Getting Started with HFSS 3D Layout: Microstrip Filter

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2024 R2
July 2024

Conventions Used in this Guide

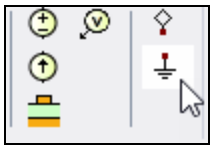
Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown (e.g., "**copy file1** " means type the word **copy**, then type a space, then type **file1**).
  - On-screen prompts and messages, names of options and text fields, and menu commands. Menu commands are often separated by greater than signs (>) (e.g., "Click **HFSS** > **Excitations** > **Assign** > **Wave Port**).
  - Labeled keys from the computer keyboard. For example, "Press **Enter** " means to press the key labeled **Enter**.
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics (e.g., "**copy***file name*" means type the word **copy**, then type a space, then type the name of the file).
- The plus sign (+) is used between keyboard keys to indicate that both keys should be pressed at the same time (e.g., "Press **Shift** +**F1** " means to press **Shift** and, while holding it down, press **F1**). Always depress the modifier key or keys first (e.g., **Shift**, **Ctrl**, **Alt**, or **Ctrl** +**Shift**), continue to hold it/them down, then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Layout** > **Interface Ground** "

  This instruction means click the **Interface Ground** command from the **Layout** tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "From the **File** menu, select **Open Examples**" means click the **File** menu and select **Open Examples** from the drop-down menu.

- Another alternative is to right-click and select from the *shortcut menu*. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means select an object, right-click, and click an option from the shortcut menu that appears.

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. This information can also be obtained by contacting an Ansys account manager.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps are taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

From the Help menu, select **Help** and choose from the following:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

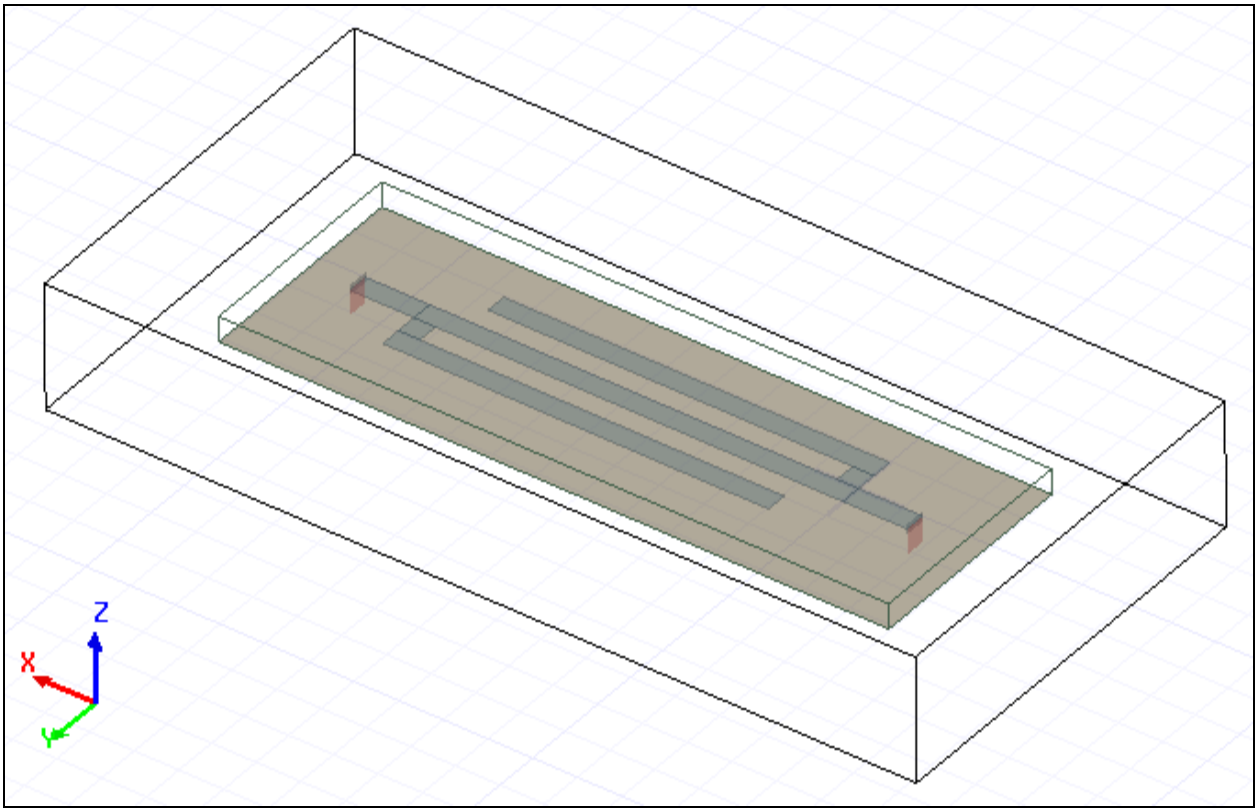
To access help from the user interface, press **F1** to open the chosen help for the active product.

Press **F1** while the cursor is pointing at a menu command or while a particular window tab is open. In this case, the help page associated with the command or open window is displayed automatically.

Table of Contents

1 - Introduction

Complete the **Getting Started with HFSS 3D Layout: Microstrip Filter** guide to create, solve, and analyze a microstrip bandstop filter using the HFSS 3D Layout design type in the **Electronics Desktop** application. A bandstop filter (also known as a band-rejection filter) is a device that attenuates signals within a target frequency band while passing higher and lower frequencies unaltered. It is the opposite of a bandpass filter.



Note:

This figure was captured with the *HFSS 3D Layout> HFSS Extents* option enabled. The outer envelope represents the boundary of the default air box surrounding the model, which is the extent of the solution region meshed and solved an HFSS analysis is performed.

In the other *Getting Started with HFSS 3D Layout* guides (*Slot Fed Patch Antenna* and *Low Pass Filter*) the ground plane layer of the stackup was defined as **Negative**, and any objects drawn on that layer represented areas of removed conductor. In this exercise, the ground layer is *not* Negative, then draw the ground plane conductor object.

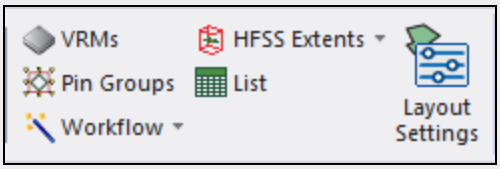
Draw the model using parametric design variables for the signal layer. Then evaluate and compare the filter response using both HFSS and Planar EM analyses.

Set General Options

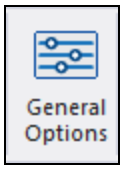
Before creating the bandstop filter, follow these steps to ensure the default unit of length measurement is set to **mil**.

Note:

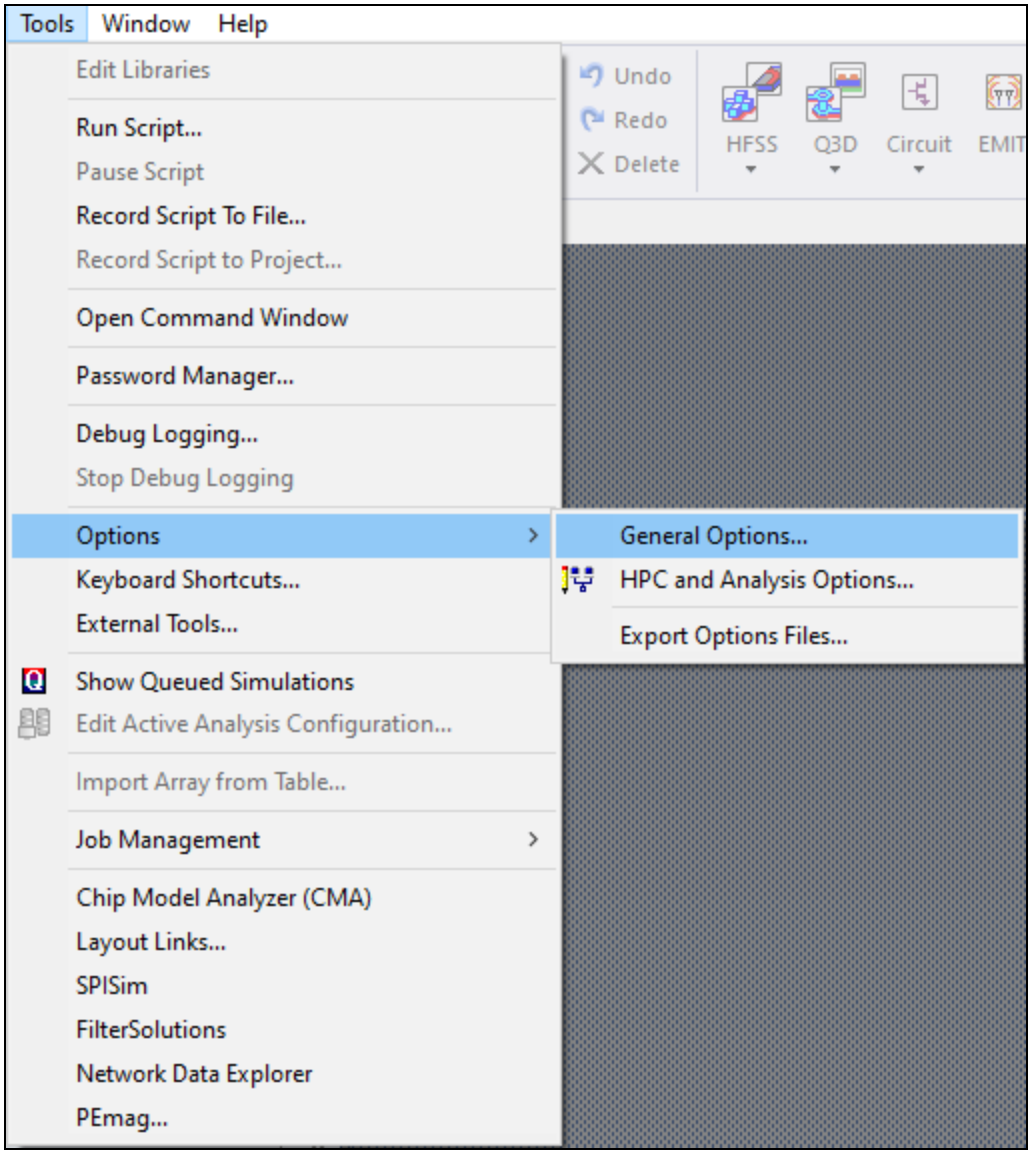
Define general settings before adding an **HFSS 3D Layout Design** element to a project. The general options control the default settings when the design type is added. To change default settings after a design has been added to the project, from the **Layout** tab, select **Layout Settings**.



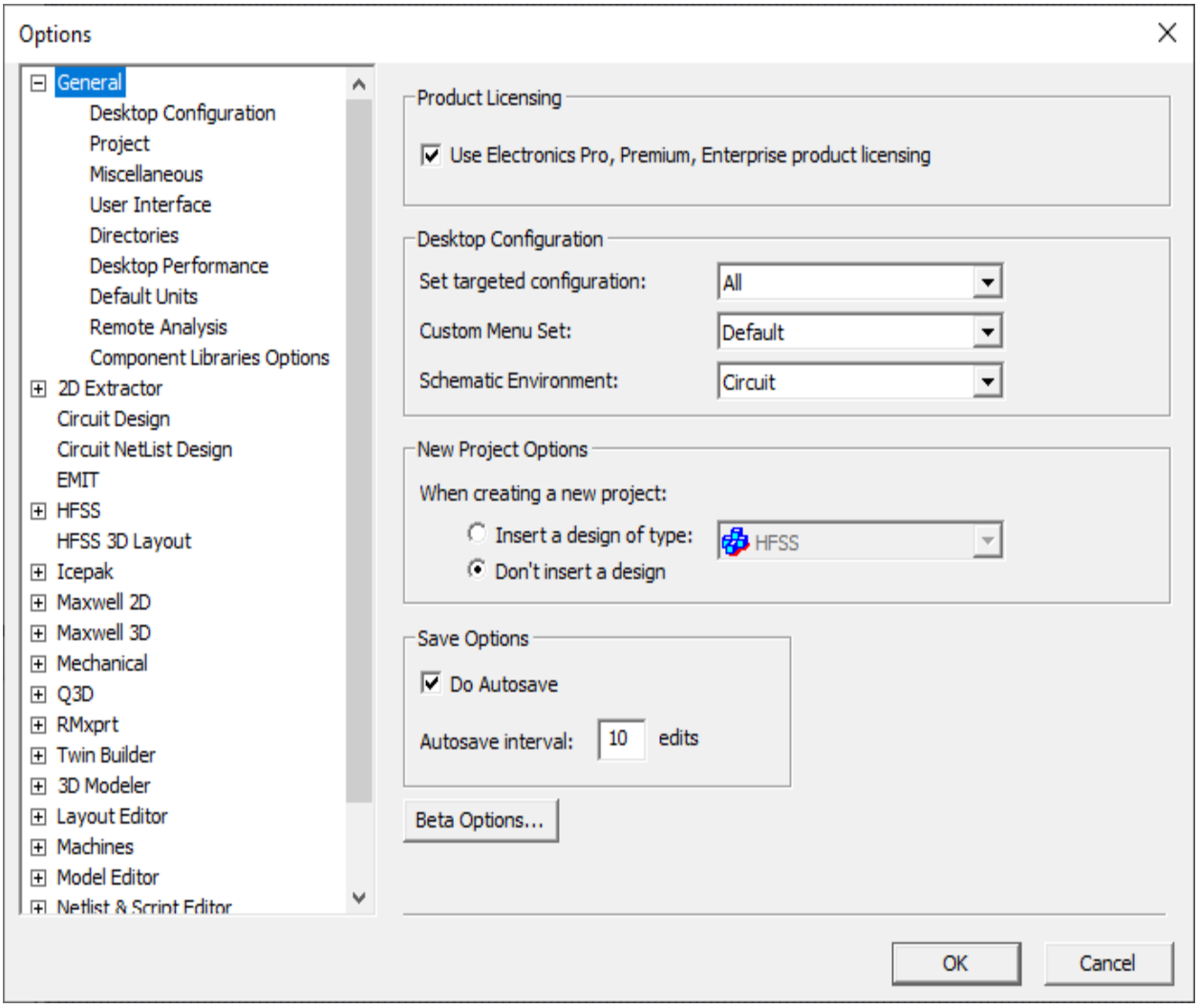
1. Open the **Options** window by doing one of the following:
    - From the **Desktop** ribbon, click **General Options**.



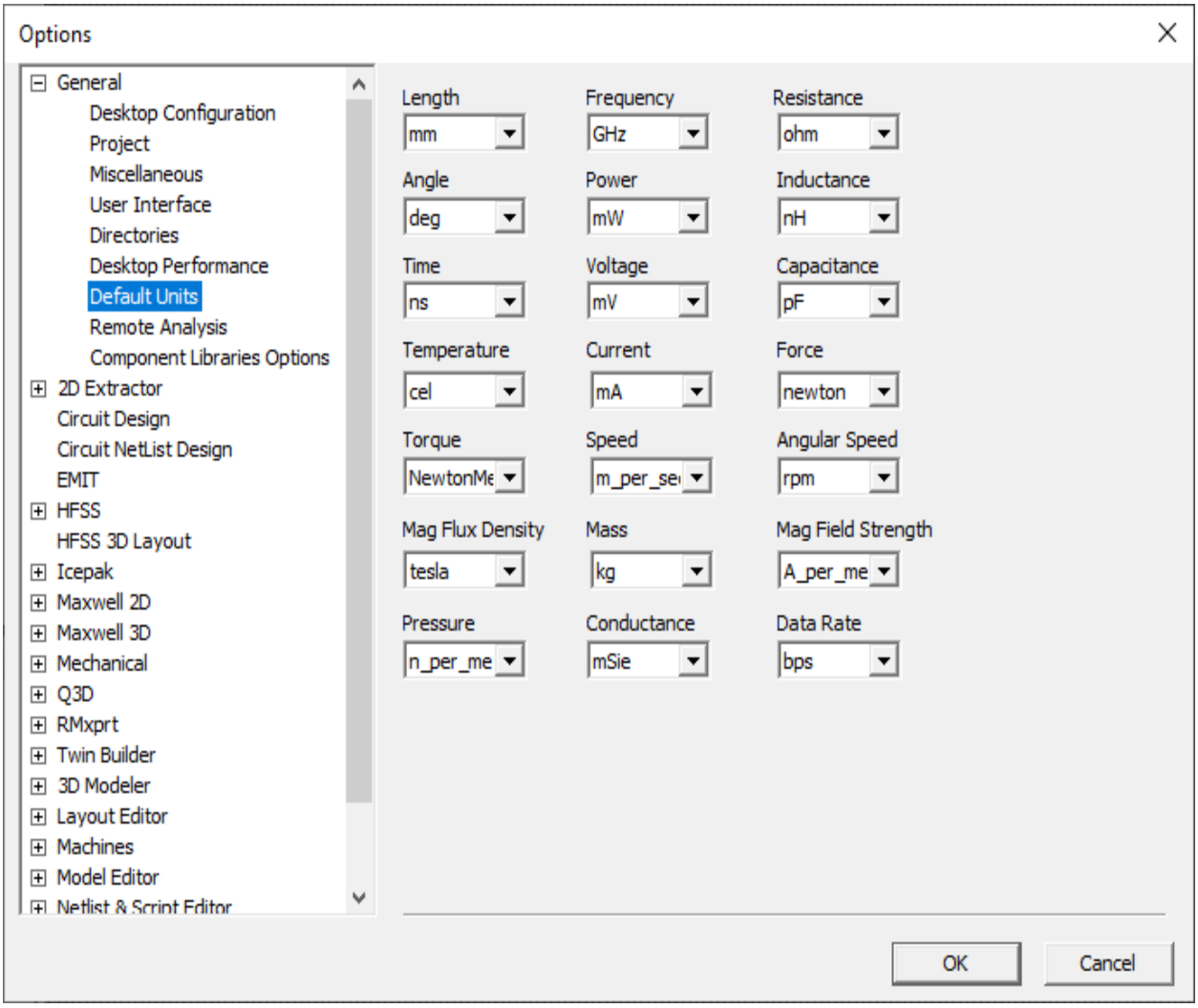
- From **Tools**, select **Options** > **General Options**.



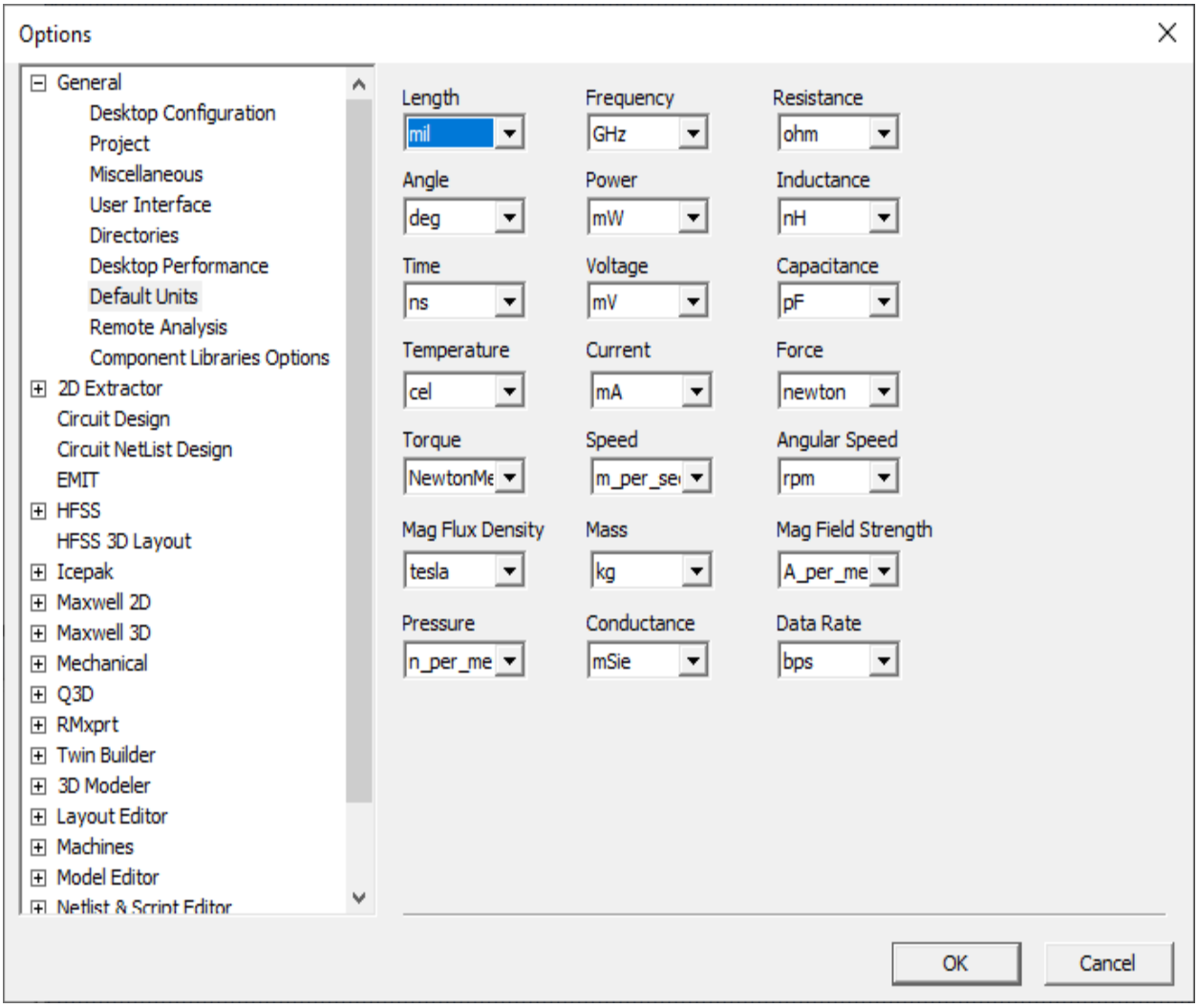
2. If appropriate, expand the **General** group.



3. Select **Default Units**.



4. Select **mil** from the **Length** drop-down menu.

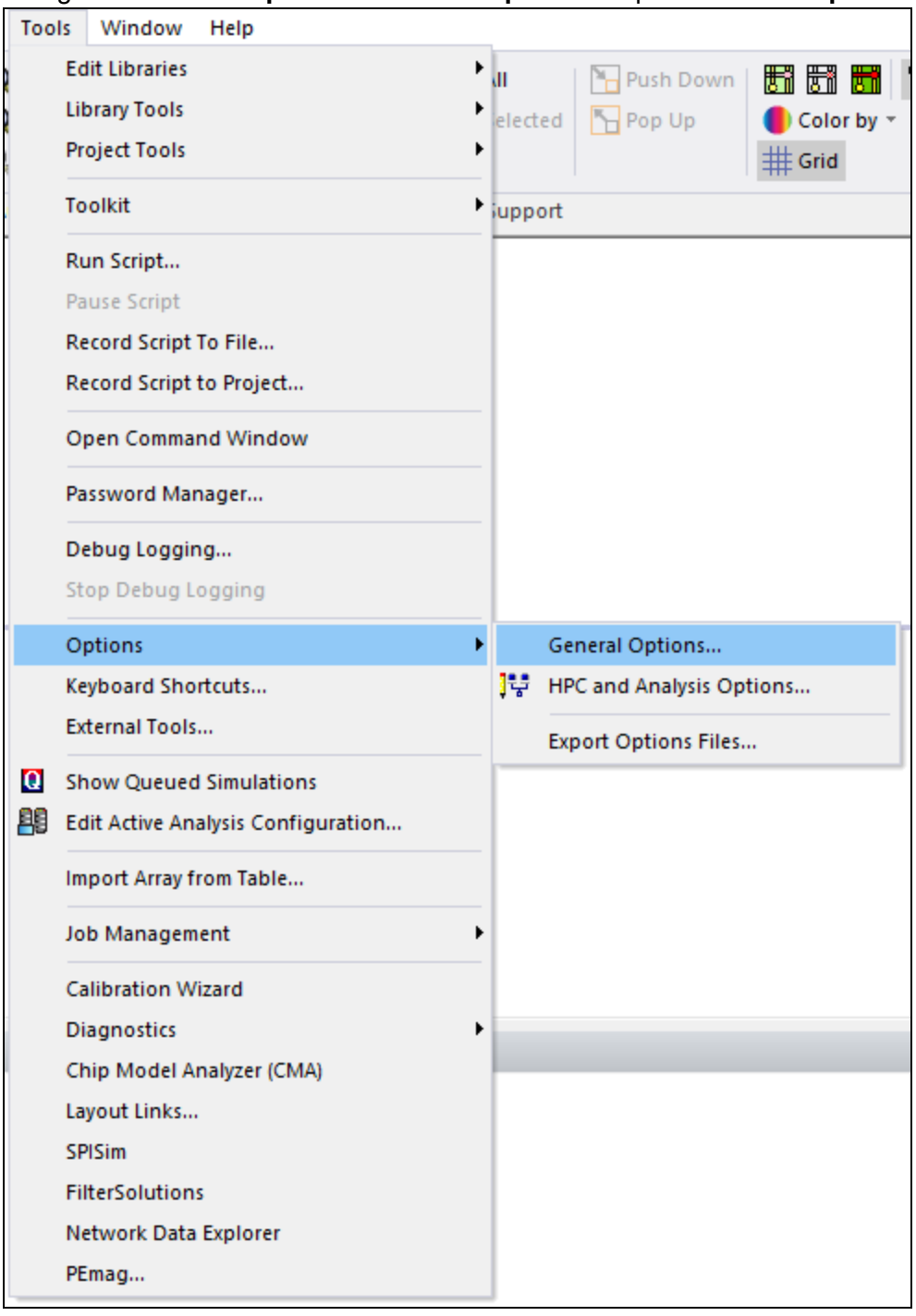


5. Click **OK** to close the **Options** window.

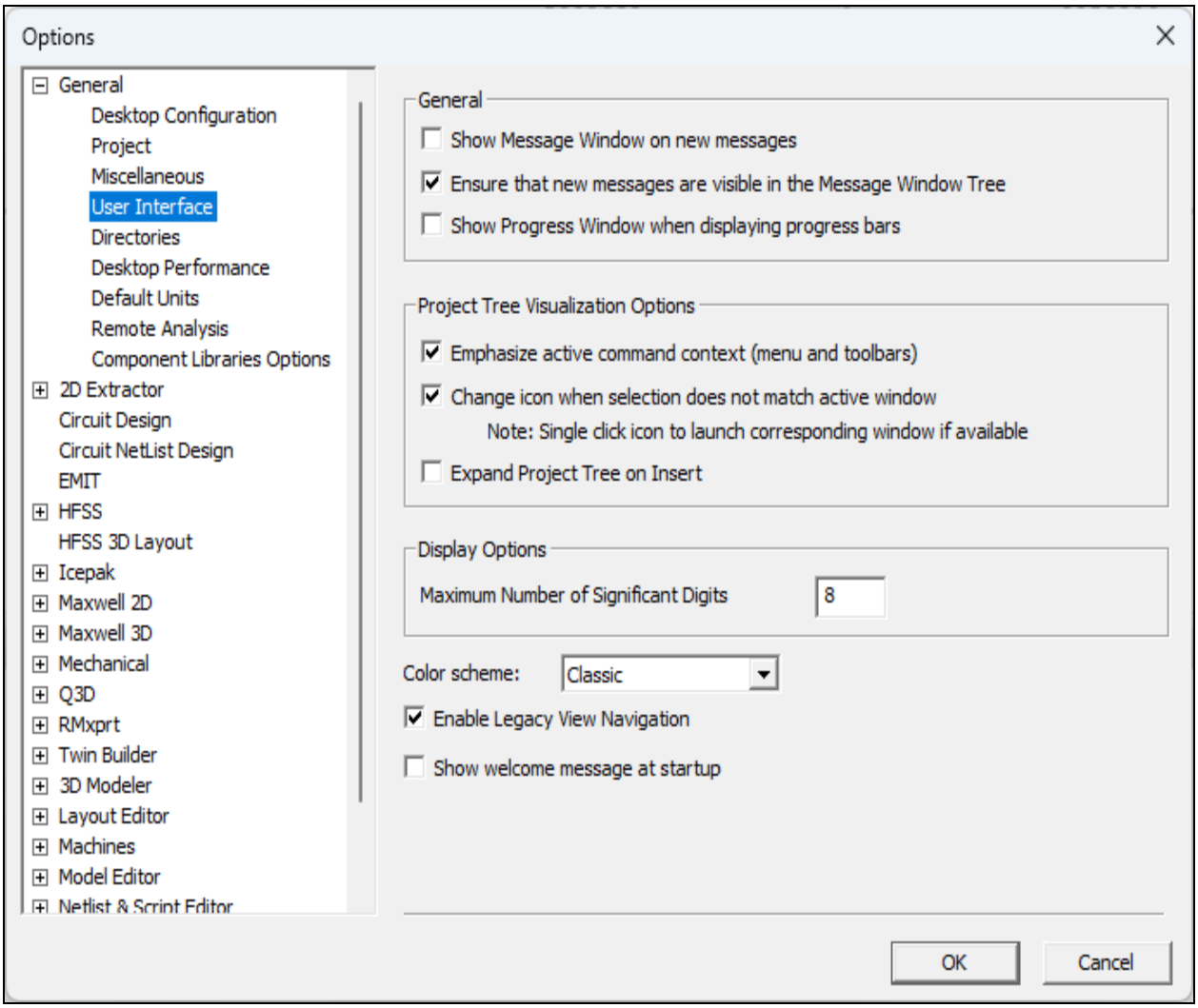
Enabling Legacy View Orientation

The instructions and examples in this guide use the legacy view orientation scheme, rather than the controls introduced in release 2024 R1. Complete these steps to enable the **Legacy View Orientation** and avoid any confusion.

1. Navigate to **Tools** > **Options** > **General Options** to open the **3D UI Options** window.



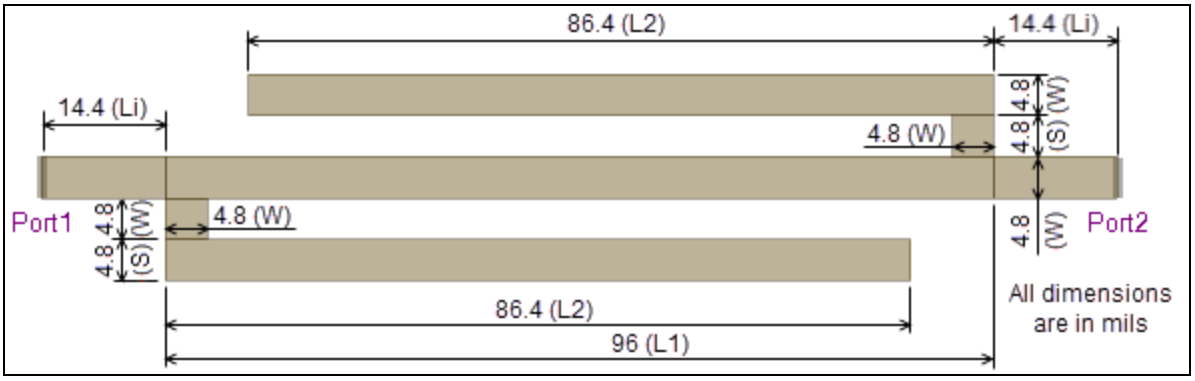
2. Expand **General** and select **User Interface**.
3. Check the **Enable Legacy View Orientation** box. When the user has completed the **Getting Started Guide**, they should return to **Options** window and uncheck the **Enable Legacy View Orientation** box.



4. Click **OK**.

Model Diagram

The following figure is a diagram of the Trace (signal) layer of the microstrip bandstop filter model:



There is also a 150 x 50 mil rectangular ground plane conductor that can be drawn on a separate layer.

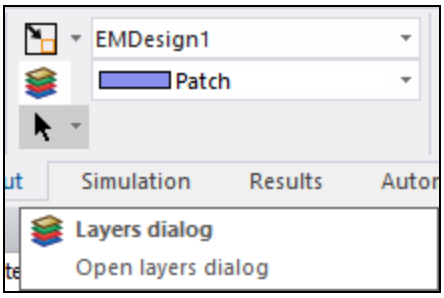
Continue to Inserting Layers.

.

Inserting Layers

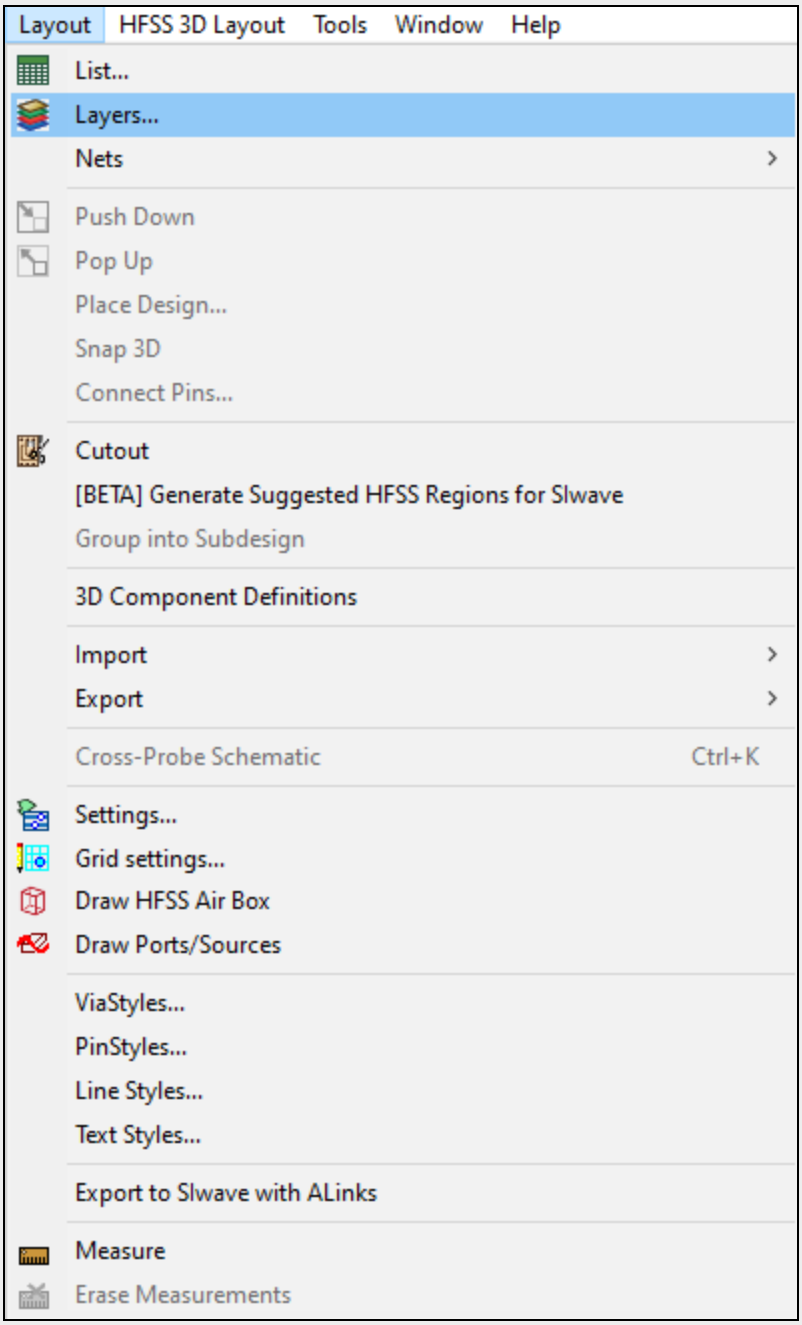
Complete these steps to insert layers in an HFSS 3D Layout design.

1. From the **Layout** tab, click the **Layers dialog** button to open the **Edit Layers** window.

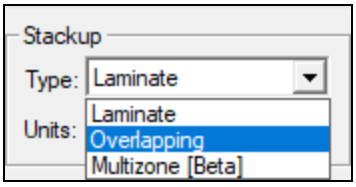


Note:

Alternatively, from **Layout**, select **Layers**.



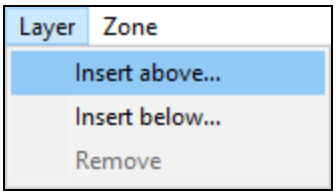
2. In the **Edit Layers** window > **Stackup** area, select **Overlapping** from the **Type** drop-down menu.



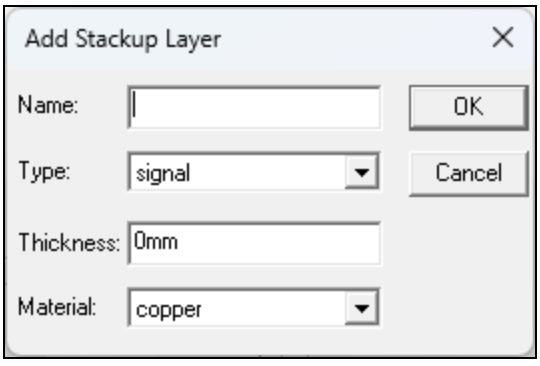
Add a Ground Layer to the Grid Control Table

> **Note:**
>
> When adding the first layer to the Grid Control Table, the actions of **Insert above** and **Insert below** are identical. Once there are one or more layers in the table, the **Insert above** and **Insert below** options will be inactive until a layer is selected from the table. After selecting a layer from the table, select the chosen option depending on where you would like the new layer to appear in the table (i.e., above or below).

1. Click **Layer** and select either **Insert above** or **Insert below** to open the **Add Stackup Layer** window.



2. In the **Add Stackup Layer** window, do the following:

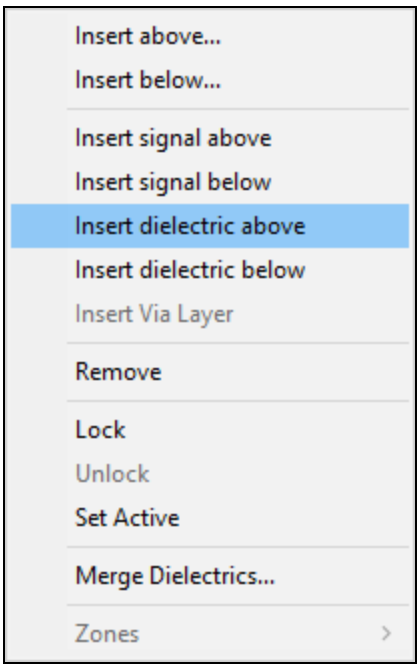


a. Enter **GND** in the **Name** field.
b. Click **OK** to close the **Add Stackup Layer** window add the new ground layer to the Grid Control Table.

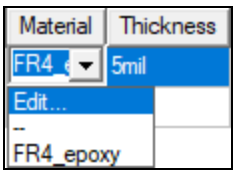
> **Note:**
>
> The material *copper* is automatically assigned to signal layers.

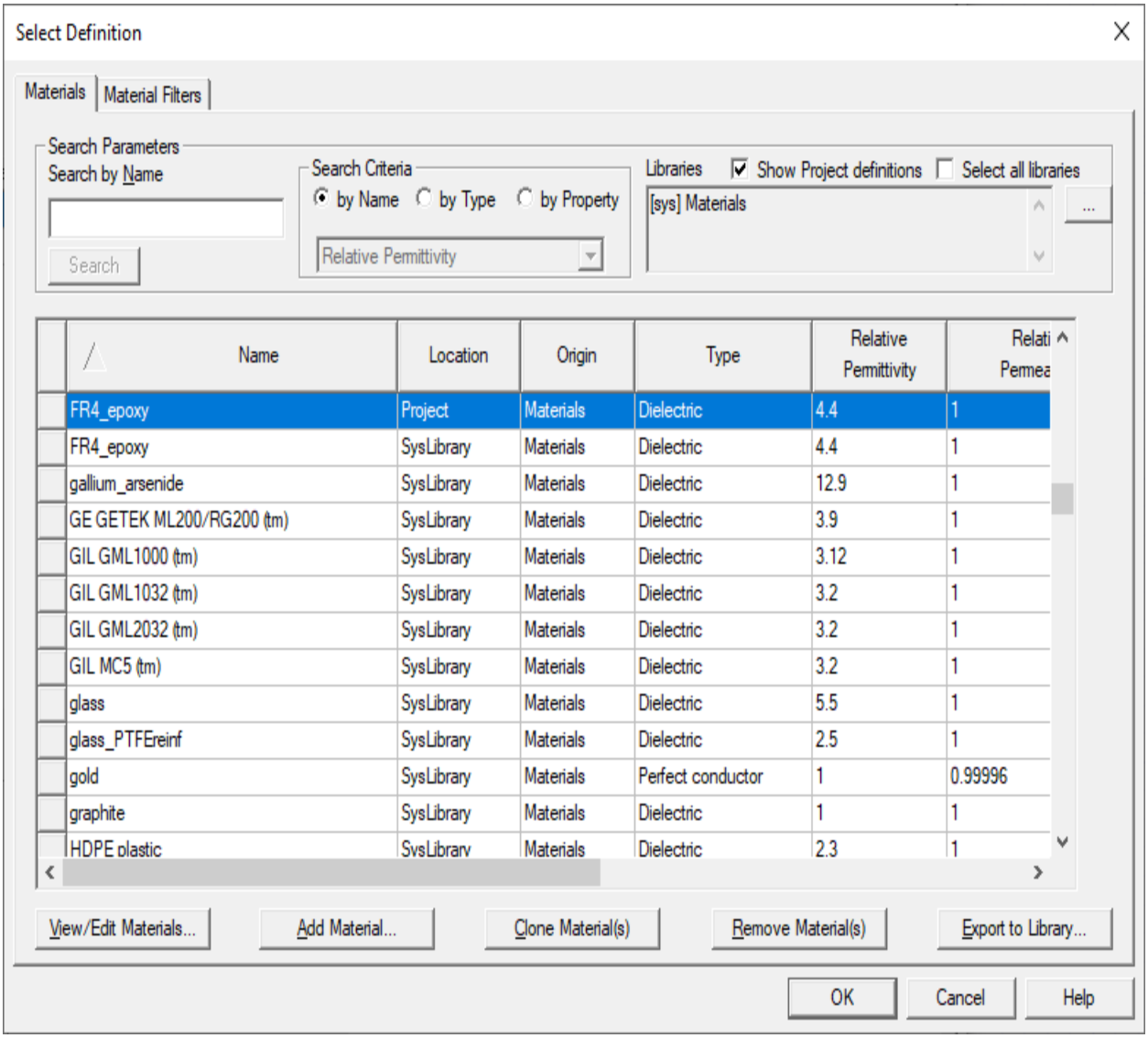
Add a Dielectric Layer to the Grid Control Table

1. Right-click anywhere in the **GND** layer and select **Insert dielectric above**. A new row appears in the Grid Control Table (default **Name**, **Dielectric**).



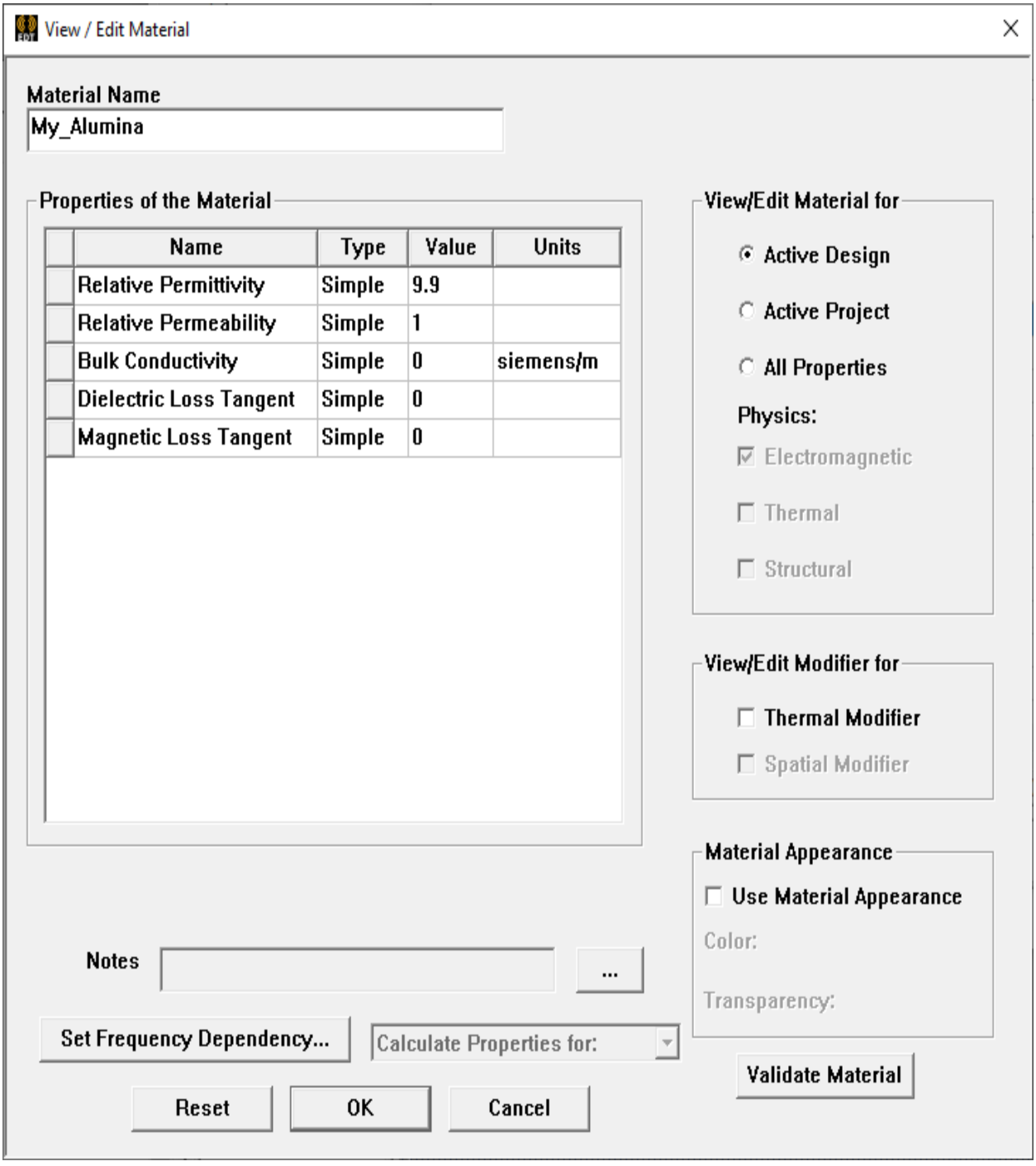
2. In the new **dielectric** row, do the following:
   a. In the **Name** field, replace **Dielectric** with **Sub1**.
   b. Ensure **5mil** is entered in the **Thickness** field.
   c. Select **Edit** from the **Material** drop-down menu to open the **Select Definition** window.





3. In the **Select Definition** window, do the following:
    a. Click **Add Material** to open the **View / Edit Material** window.
    b. Enter **My_Alumina** in the **Material Name** field.
    c. Enter **9.9** in the **Relative Permittivity** field.

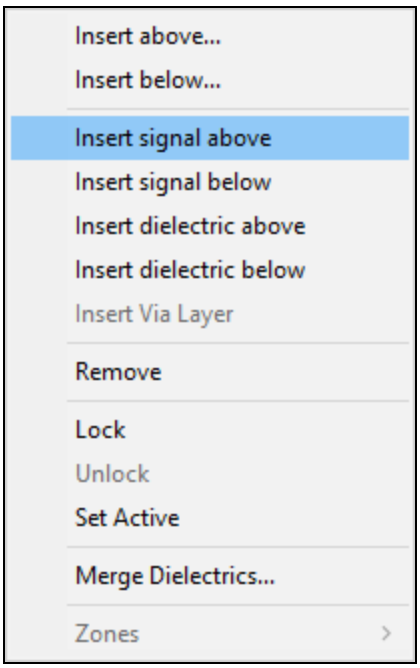
d. Click **OK** to save changes, close the **View / Edit Material** window, and return to the **Select Definition** window.



e. Click **OK** to close the **Select Definition** window.

Add a Trace (Signal) Layer to the Layer Grid Control Table

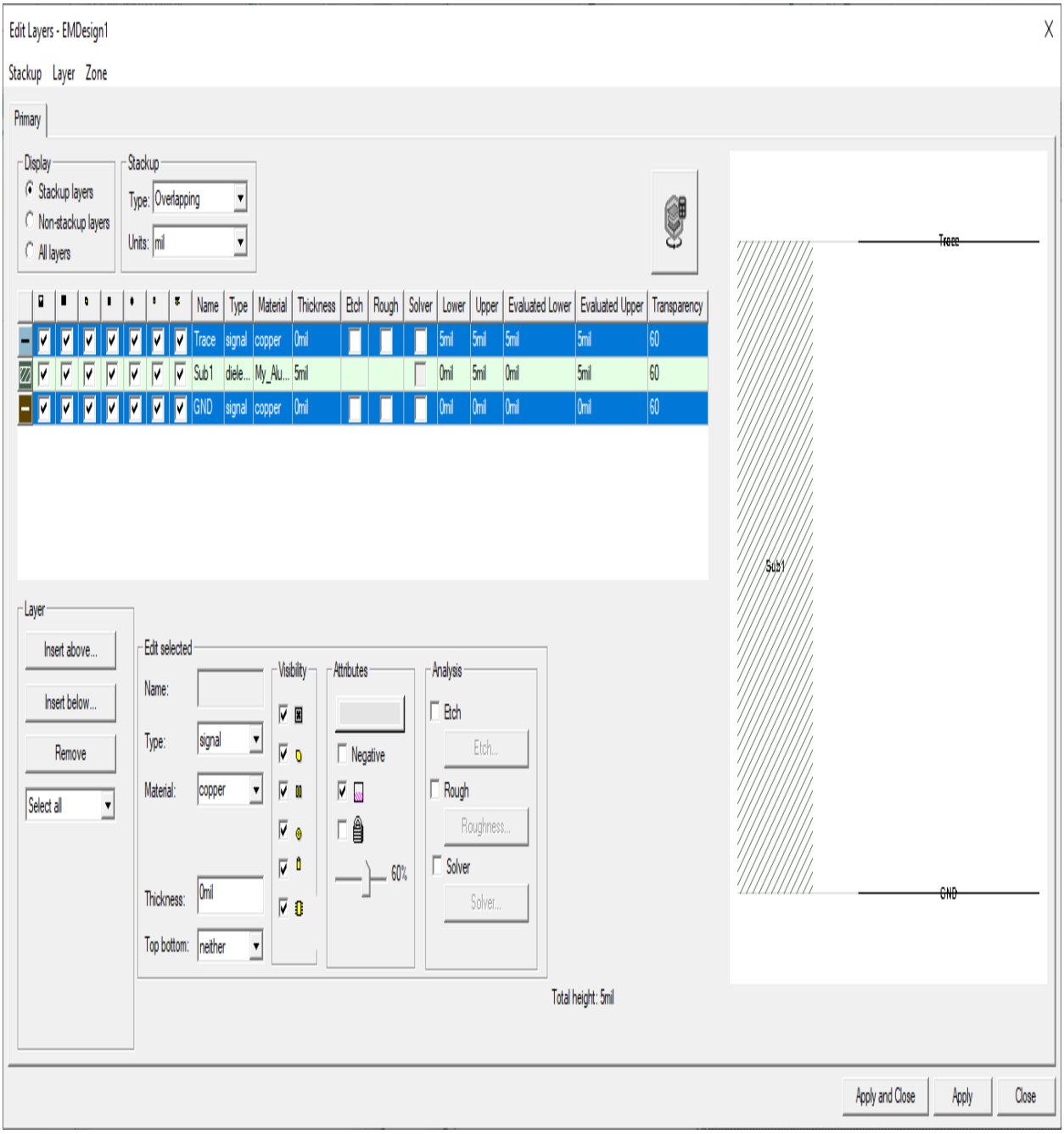
1. Right-click the **Sub1** layer and select **Insert signal above**. A new row appears in the Grid Control Table.



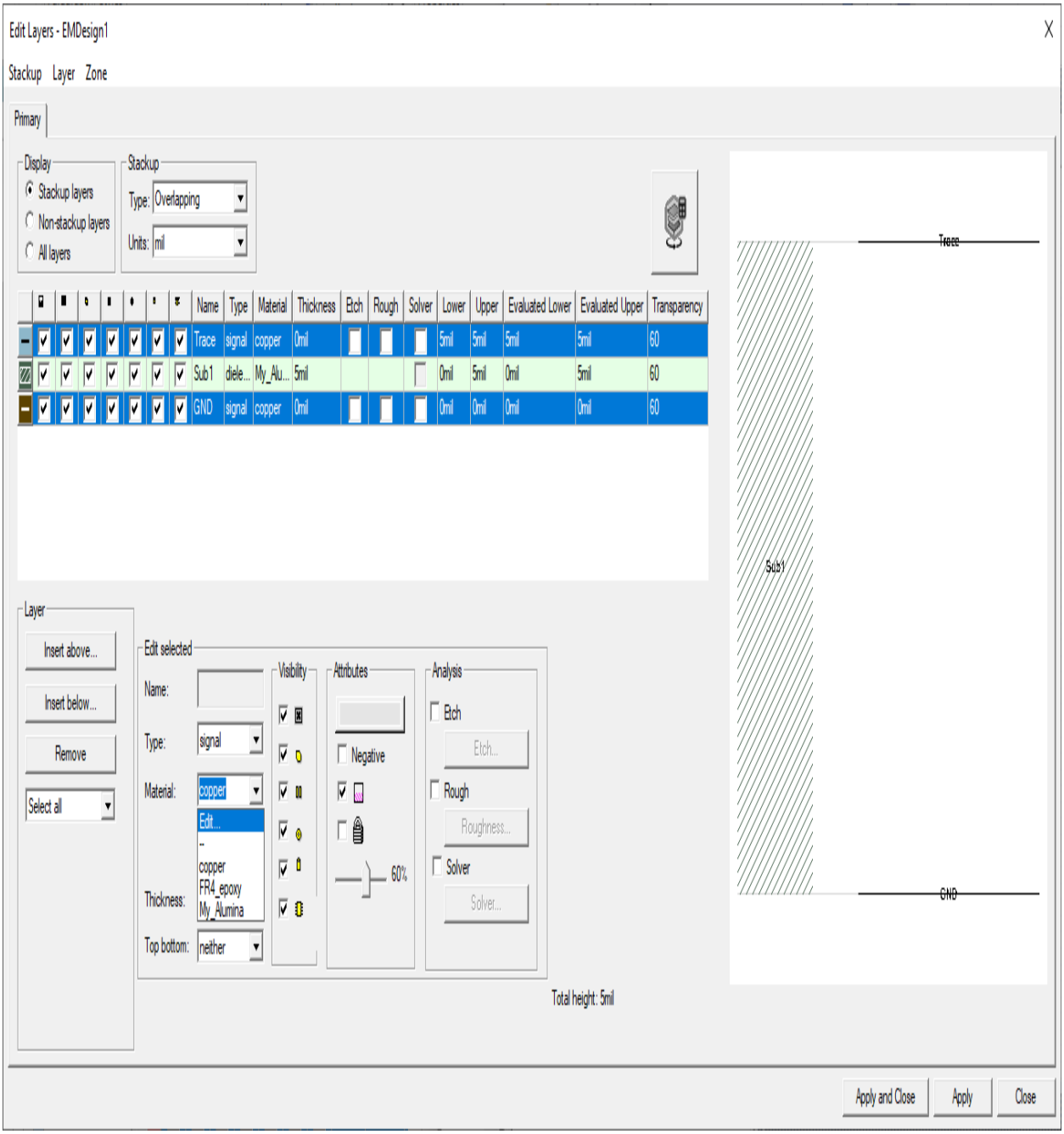
2. In the new **signal** row, replace **Signal** in the **Name** field with **Trace**.

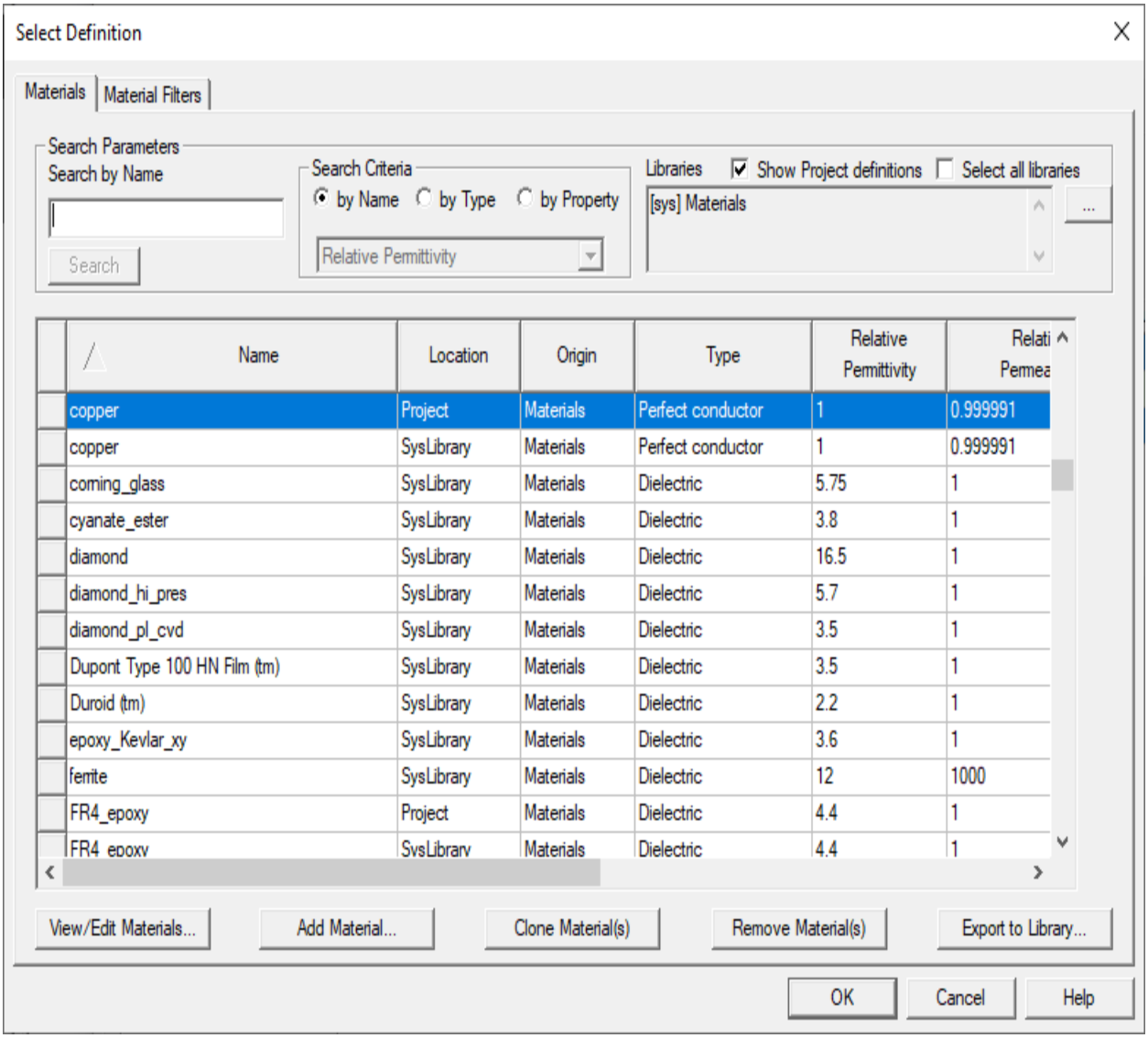
Make Changes to All Layers

1. With the **Trace** layer selected, hold **Ctrl** and click the first field in the **GND** row. Both layers should be highlighted (i.e., selected).

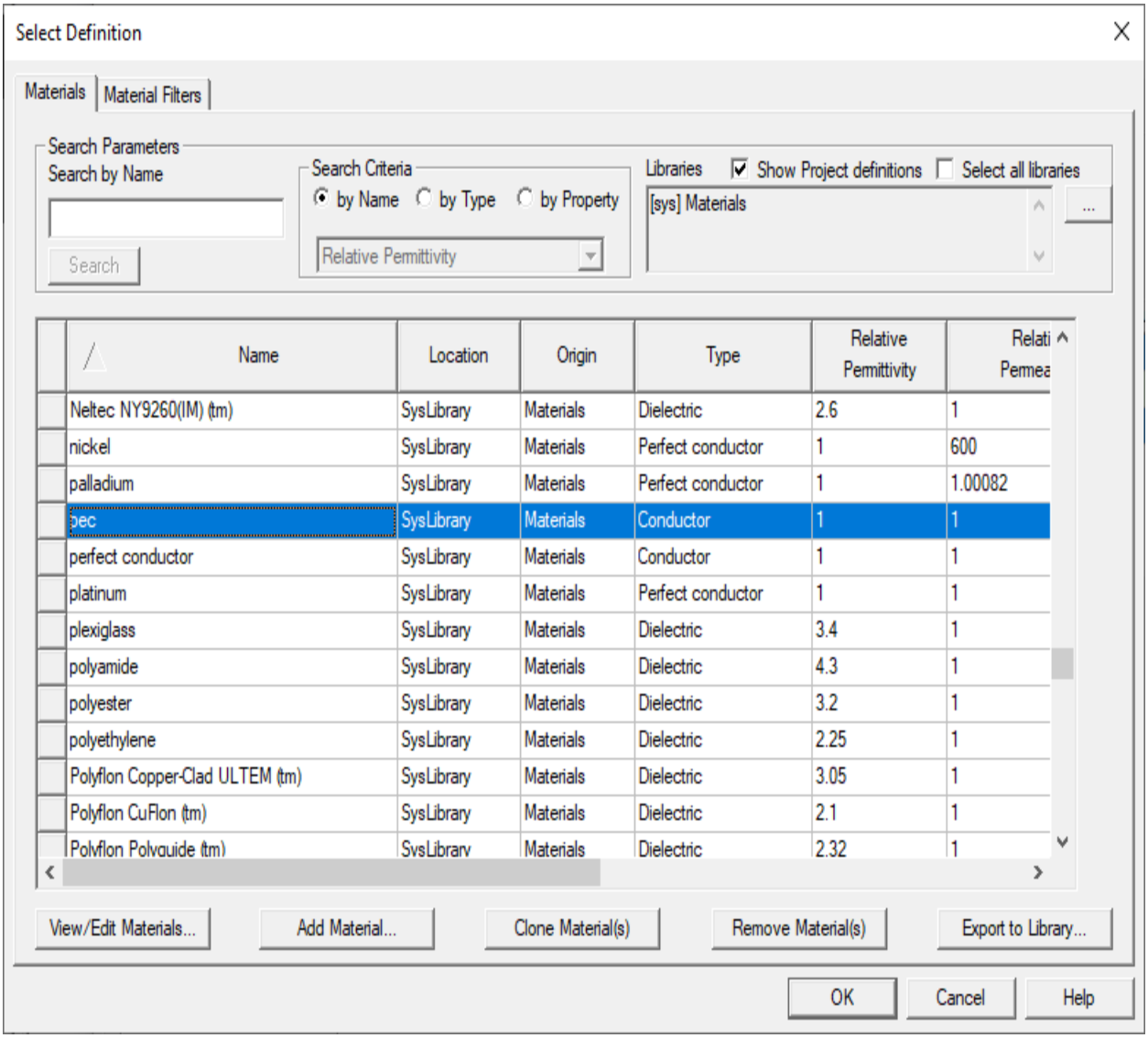


2. From the **Edit selected** area, select **Edit** from the **Material** drop-down menu to open the **Select Definition** window.

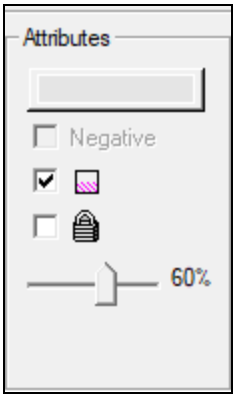




3. Select **pec** (perfect electrical conductor) from the list of library materials. Then click **OK** to close the **Select Definition** window.



4. From the **Layer** area, choose **Select all** from the drop-down menu.
5. Ensure the "shading" box in the **Attributes** area (i.e., the middle box) is checked. This ensures that all objects will be shaded, rather than only outlined (i.e., wire frame).



Note:

Checking the shading" box ensures that all the objects drawn on each layer is colored in, as opposed to appearing as only an outline.

6. The **Edit Layers** window should now match the following example.

> **Note:**
>
> If the stackup is not arranged in the correct hierarchy, rearrange the layers by **clicking+dragging** the selection handles in the left column. The **t1** layer should be from the top of the list, followed by the **d1** layer in the middle, and the **g1** layer from the bottom.

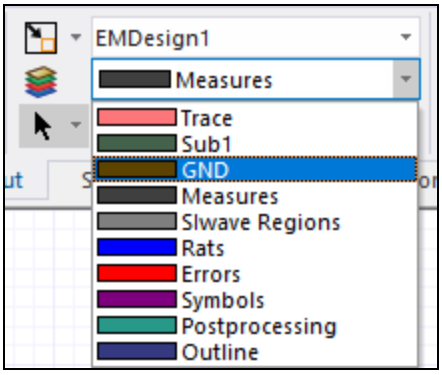
7. Click **Apply and Close** to apply the layer definitions and close the **Edit Layers** window.

Continue to Drawing the Ground Plane.

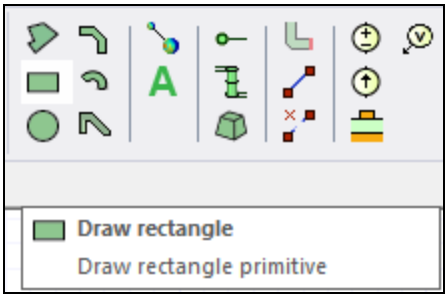
Drawing the Ground Plane

Complete these steps to draw a ground plane in the **Layout Editor**.

1. From the **Layout** tab, select **GND** from the **Active Layer** drop-down menu.



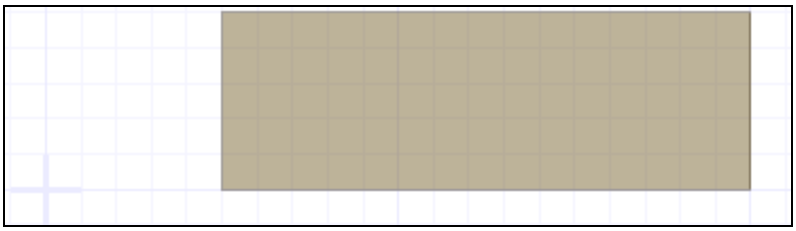
2. From the **Layout** tab, click **Draw rectangle**.



3. Do **not** **click+drag** in the **Layout Editor**. Instead, move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Enter **50** in the field.

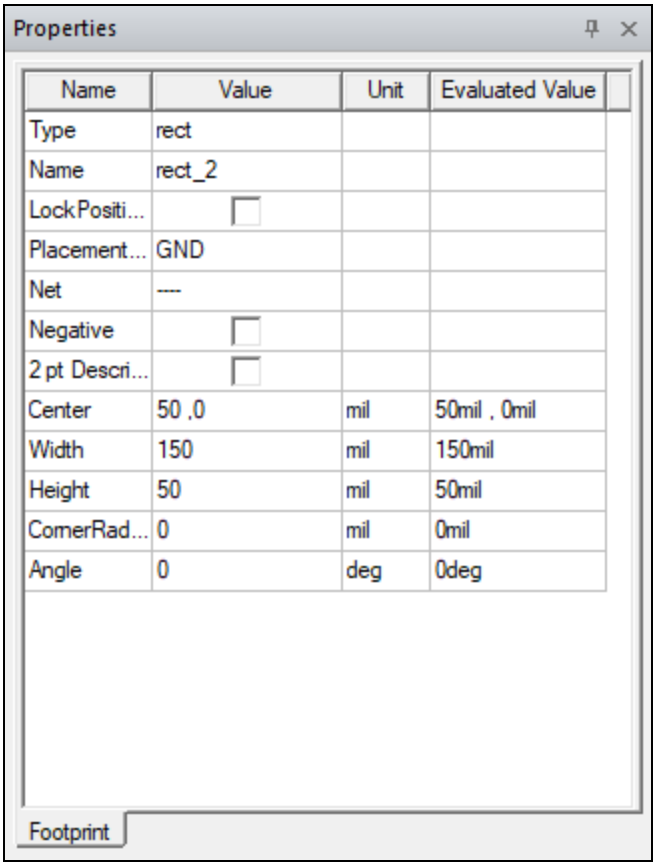
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.

5. Either press **Tab** until the cursor moves to the **Delta X** coordinate field or move the cursor to the field, click inside it, and enter **150**.
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field, Then type **50** in the field and press **Enter** to complete the ground plane.



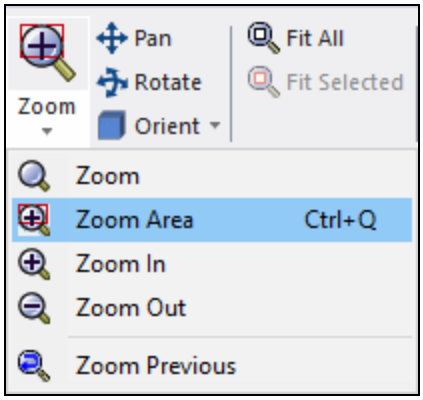
> **Note:**
>
> Coordinates entered via the coordinate fields specify opposite corners of a rectangle. Alternatively, entering the **Center**, **Width**, and **Height** of the proposed model in the **Properties** window determines the dimensions of the model dependent from the rectangle's centroid.

7. With the new ground plane selected, ensure the **2 pt Description** option in the **Properties** window is **not** selected.
8. Enter **50 ,0** in the **Center** field and press **Enter** or **Tab** to relocate the ground plane, centered at **50, 0**.

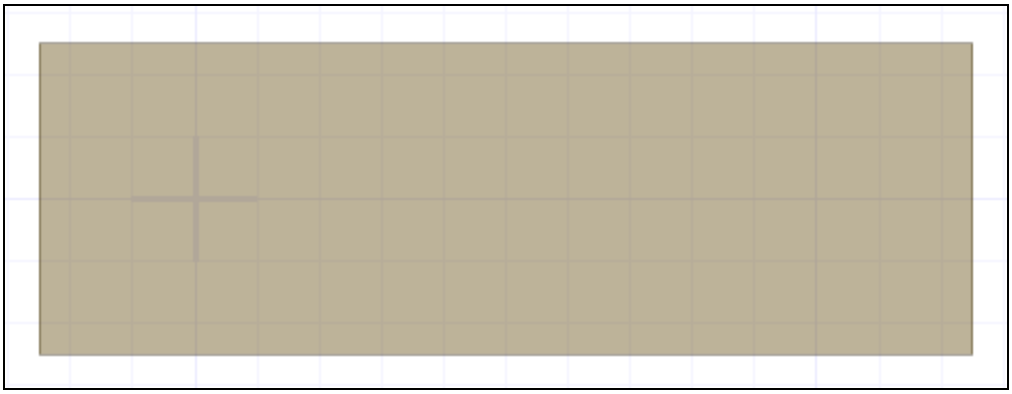


9. Click anywhere else in the **Layout Editor** to clear the current selection.
10. **Zoom In** from the space between the right edge of the first rectangle, the top edge of the second rectangle, and the left edge of the third rectangle, by doing one of the following:
    - Spin the mouse wheel to **Zoom In**/**Out**.
    - Press **Ctrl**+**D**.
    - From the **Layout** tab, click **Fit All**.
    - From **View**, select **Fit All**.

- From the Layout ribbon, select **Zoom** > **Zoom Area**. Then **click+drag** the mouse to define an area.



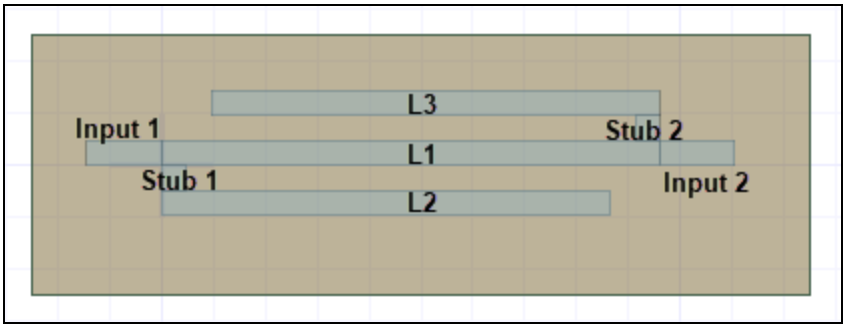
11. Click in the **Layout Editor**'s background area to clear the selection. If appropriate, press **Ctrl+D** to fit the view.



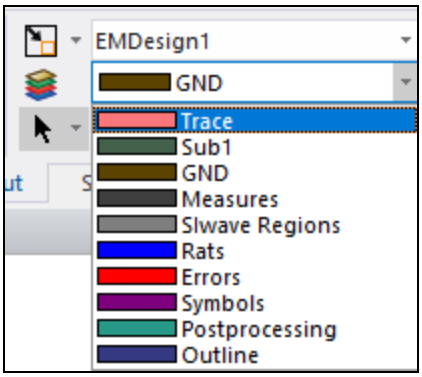
Continue to Creating L1.

Creating L1

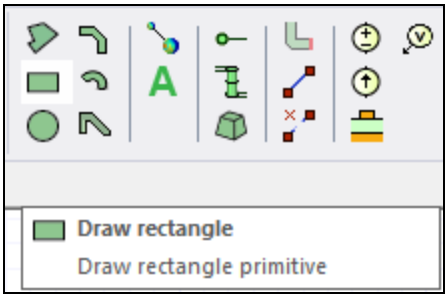
Complete these steps to create the object L1 and then parameterize the object. When an object is parameterized, variables are defined for the coordinates and dimensions rather than absolute numeric values. In this way, define additional objects based on parameters of previously defined ones, to quickly alter the geometry of the model by editing the design parameters. All geometry directly or indirectly based on an altered parameter is automatically updated.



1. From the **Layout** tab, select **Trace** from the **Active Layer** drop-down menu:

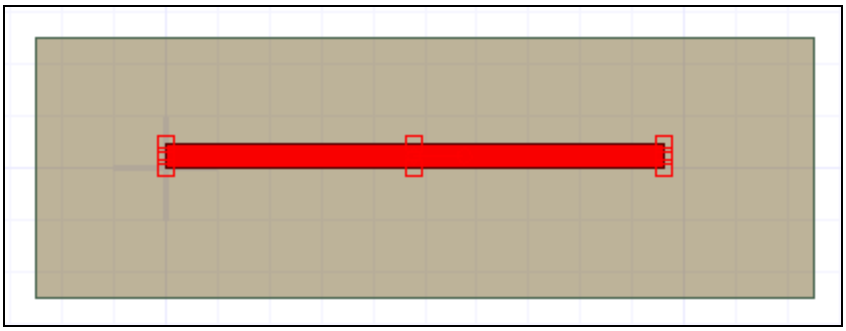


2. From the **Layout** tab, click **Draw rectangle**.



3. Do **not** **click+drag** in the **Layout Editor**. Instead, move the cursor to the **X** coordinate field at the bottom of the **Layout Editor**. Then enter **0** in the field.
4. Press **Tab** to move the cursor to the **Y** coordinate field. Then type **0** in the field and press **Enter**.
5. Either press **Tab** until the cursor moves to the **Delta X** coordinate field or move the cursor to the field, click inside it, and enter **96**.

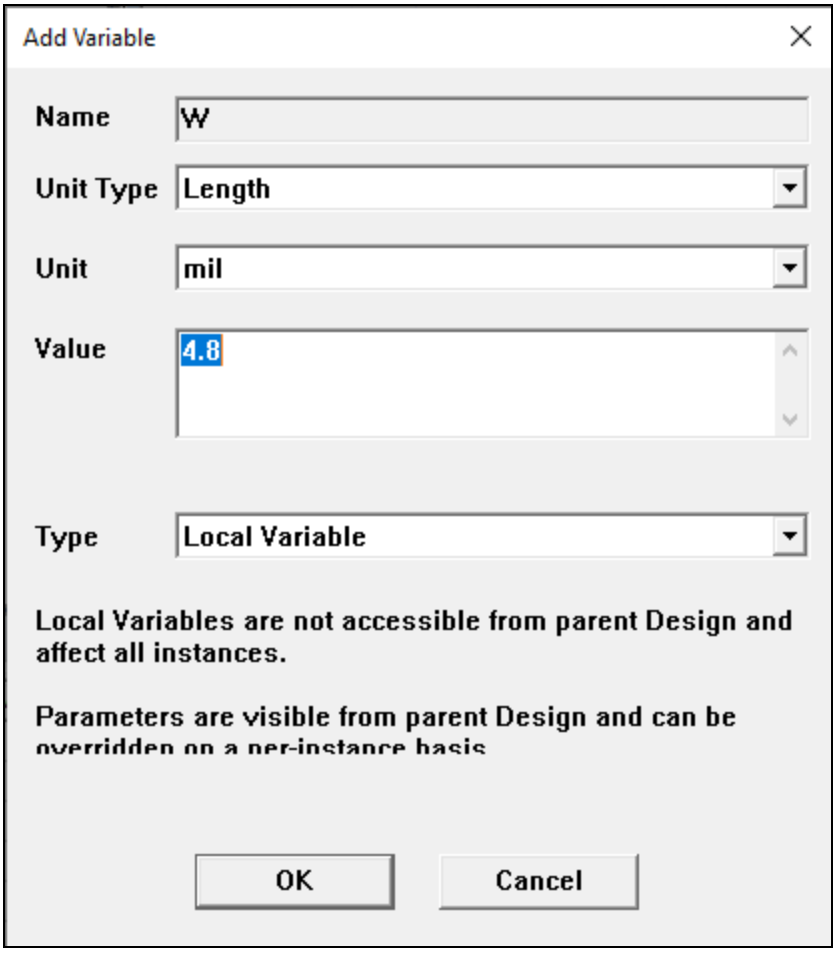
6. Press **Tab** to move the cursor to the **Delta Y** coordinate field, Then type **4.8** in the field and press **Enter** to complete the ground plane.



7. Select the new object (i.e., **L1**) to populate the **Properties** window. Then make the following changes:

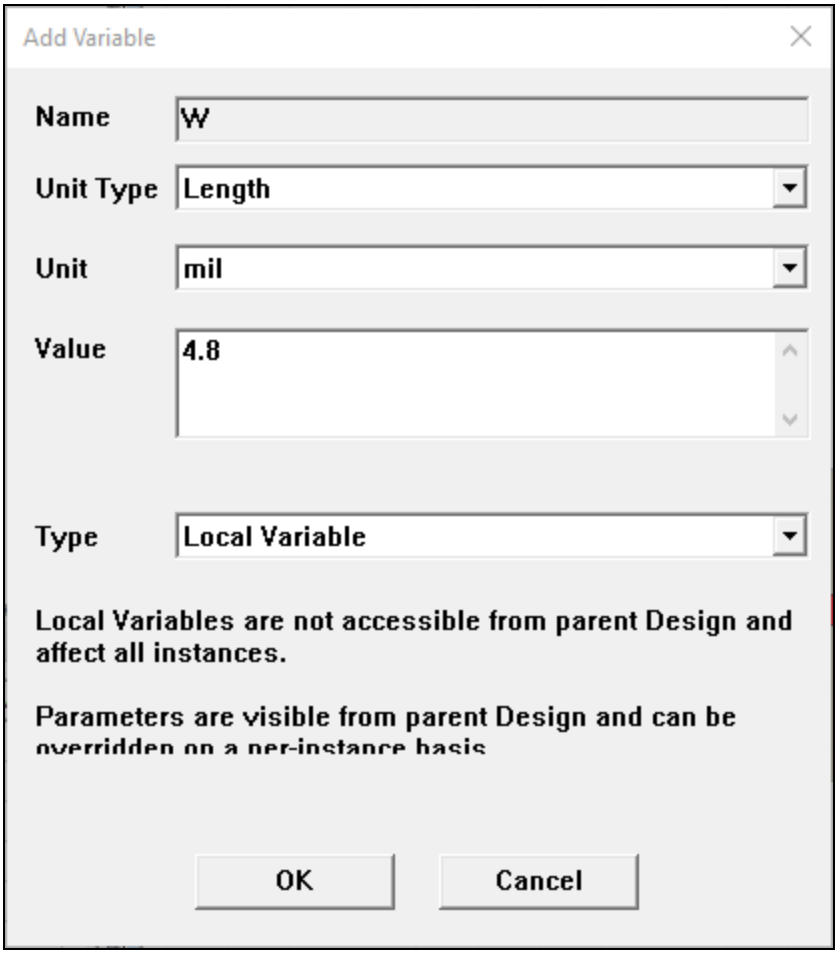
    a. Ensure the **2 pt Description** option is **not** selected.

b. Enter **L1** in the **Width** field. Then press **Enter** or **Tab** to open the **Add Variable** window.



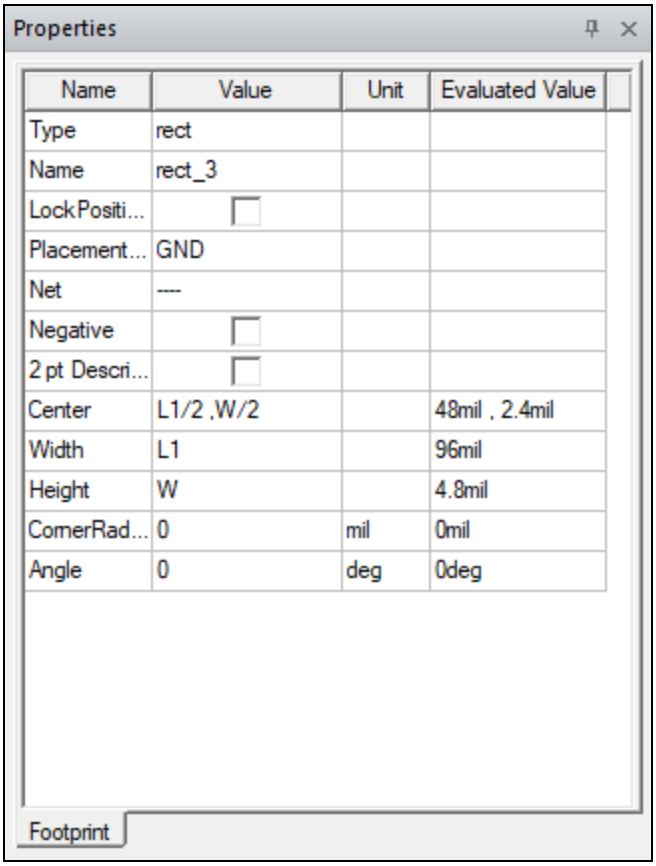
c. Ensure **mil** is selected from the **Unit** drop-down menu and **96** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.

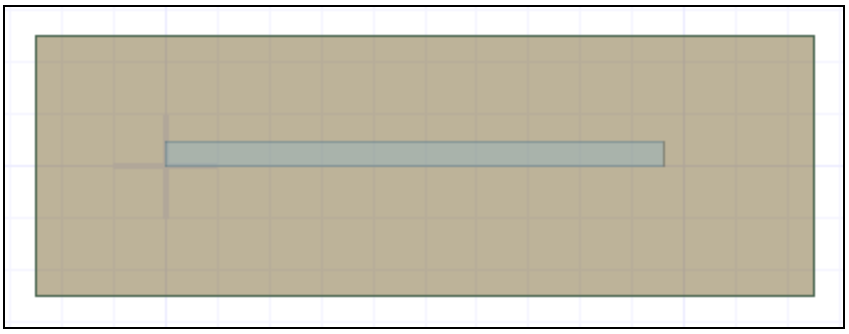
d. Enter **W** in the **Height** field. Then press **Enter** or **Tab** to open the **Add Variable** window.



e. Ensure **mil** is selected from the **Unit** drop-down menu and **4.8** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.

f. Enter **L1/2, W/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates.

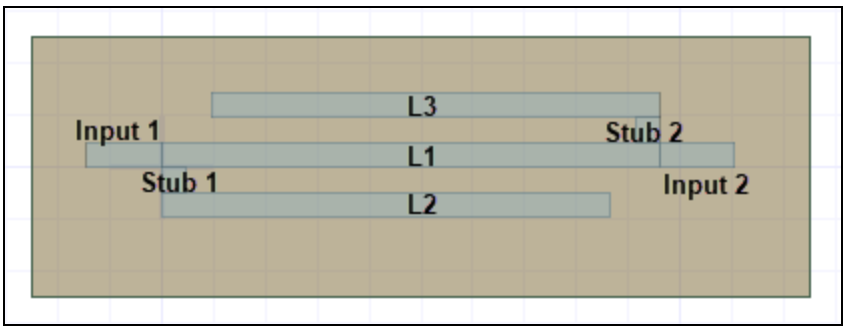




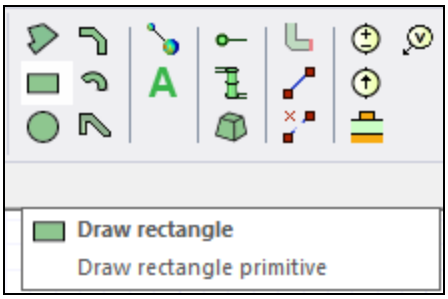
Continue to [Creating Stub 1](#).

Creating Stub 1

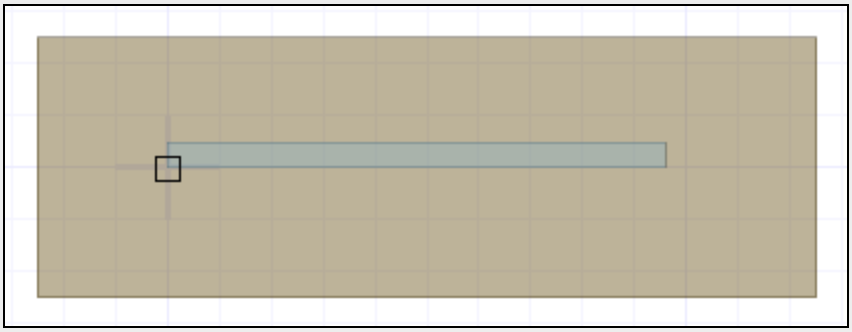
Complete these steps to create the object Stub 1 and then parameterize the object.



1. From the **Layout** tab, click **Draw rectangle**.

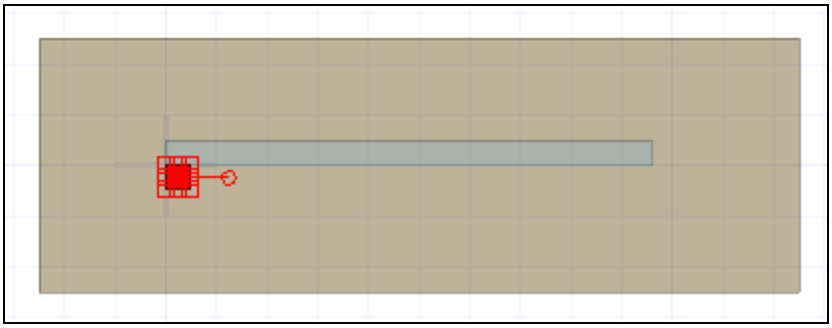


2. Click the lower-left corner of **L1** (i.e., **0, 0**).

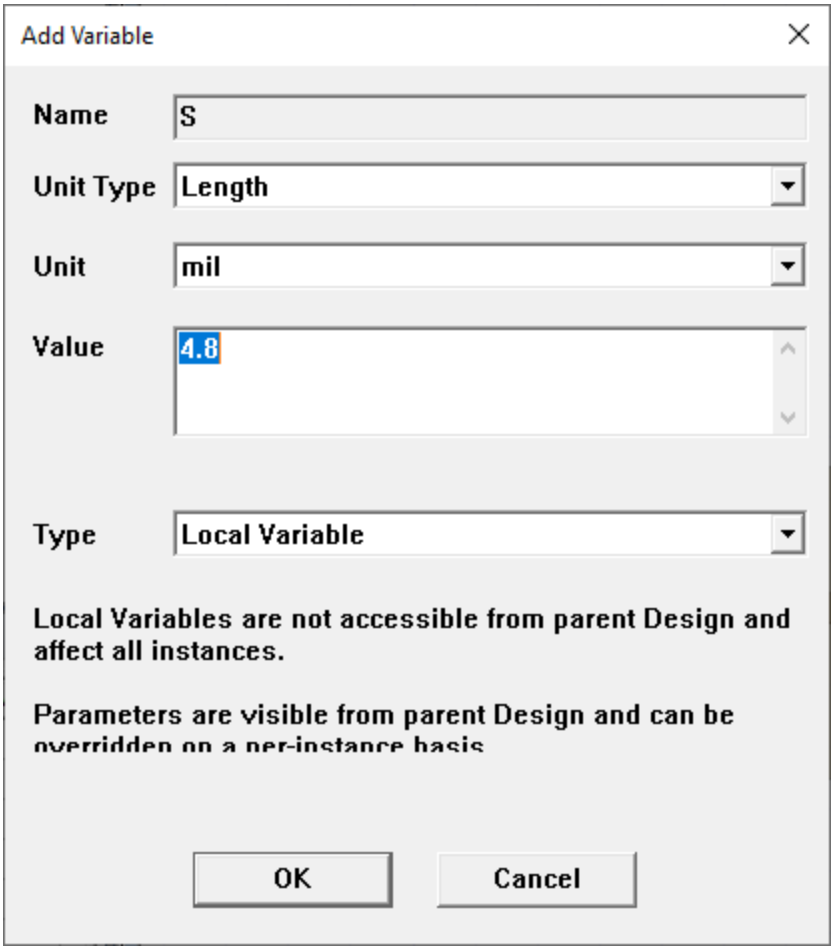
> **Note:**
>
> The cursor changes from a rectangle to a square when a snapping point is detected.
>
> 

3. Press **Tab** to move the cursor to the **Delta X** coordinate field at the bottom of the **Layout Editor**. Then enter **4.8** in the field..

4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **-4.8** in the field and press **Enter** to complete the new rectangle.

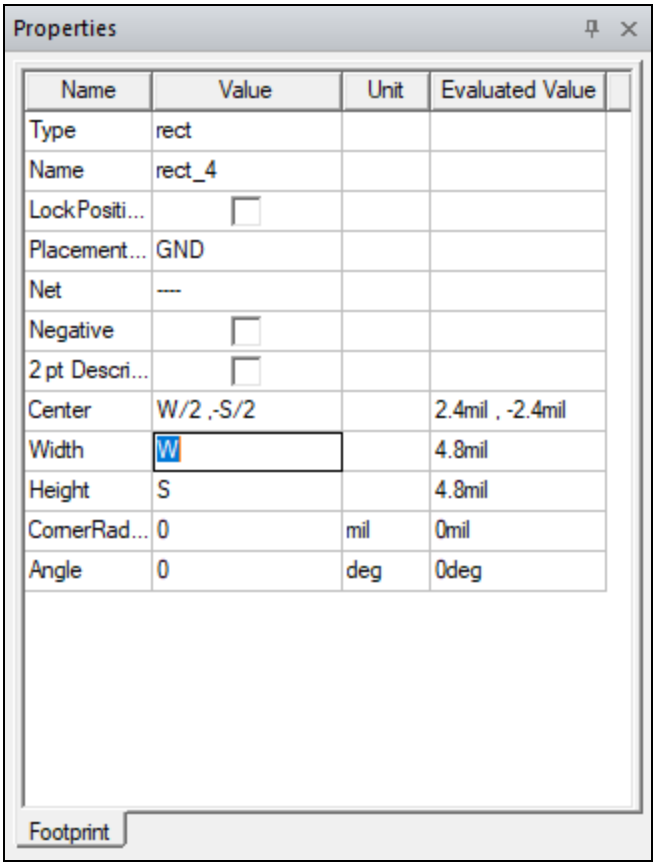


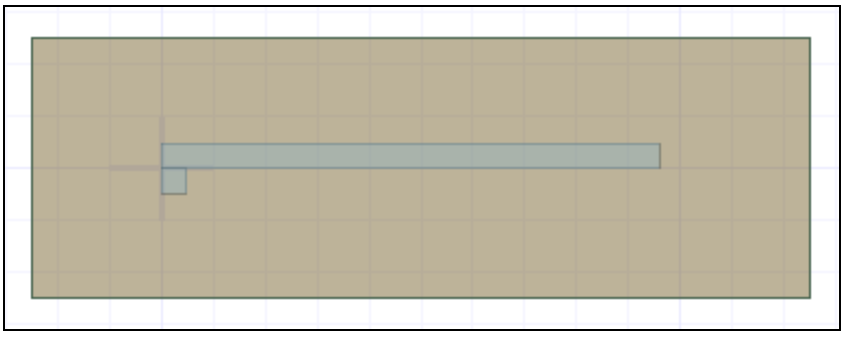
5. Select the new object (i.e., **Stub 1**) to populate the **Properties** window. Then make the following changes:
   a. Ensure the **2 pt Description** option is **not** selected.
   b. Enter **W** in the **Width** field. Then press **Enter** or **Tab**.
   c. Enter **S** in the **Height** field. Then press **Enter** or **Tab** to open the **Add Variable** window.



d. Ensure **mil** is selected from the **Unit** drop-down menu and **4.8** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.

e. Enter **W/2, -S/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates.

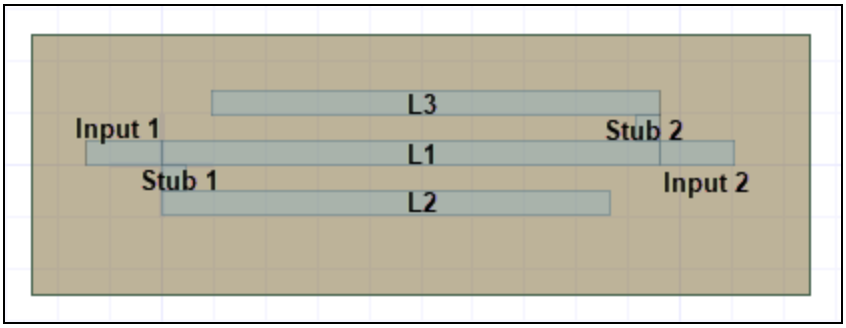




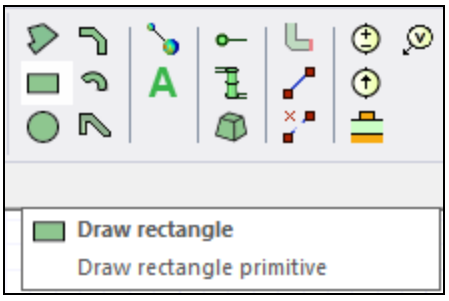
Continue to [Creating L2](#).

Creating L2

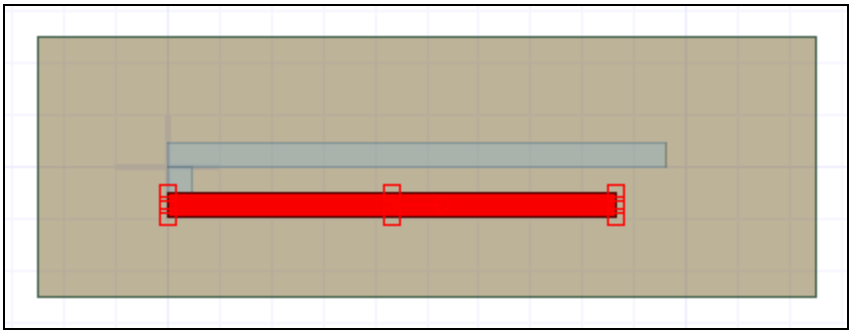
Complete these steps to create the object L2 and then parameterize the object.



1. From the **Layout** tab, click **Draw rectangle**.

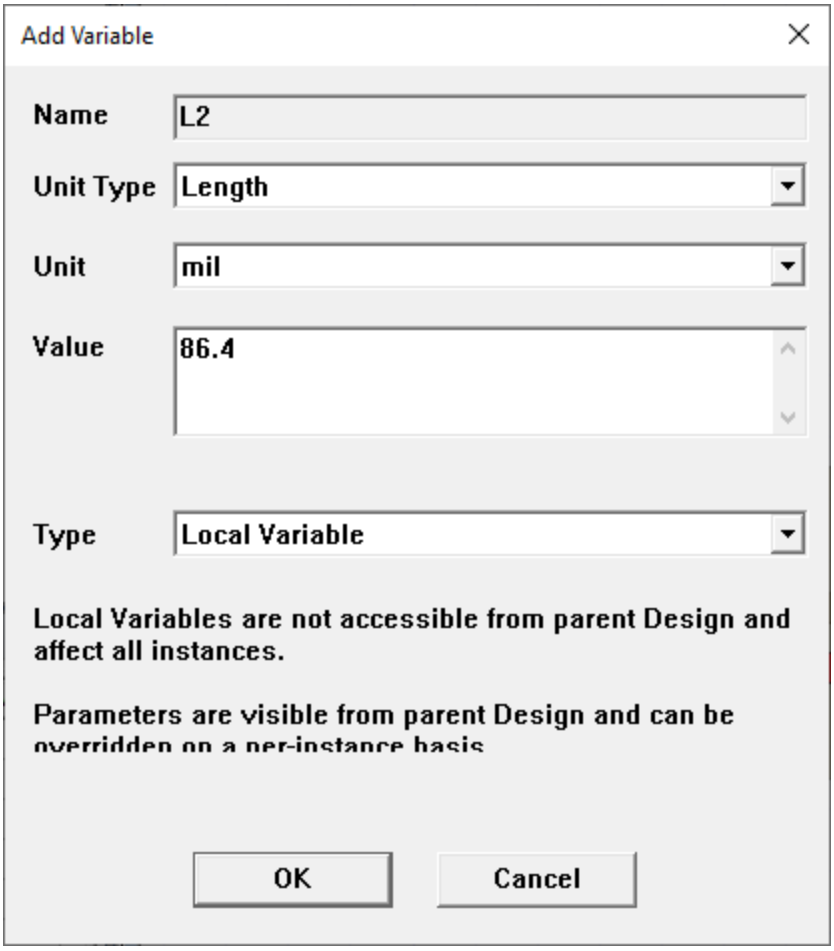


2. Click the lower-left corner of **Stub 1** (i.e., **0, -4.8**).
3. Press **Tab** to move the cursor to the **Delta X** coordinate field at the bottom of the **Layout Editor**. Then enter **86.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **-4.8** in the field and press **Enter** to complete the new rectangle.



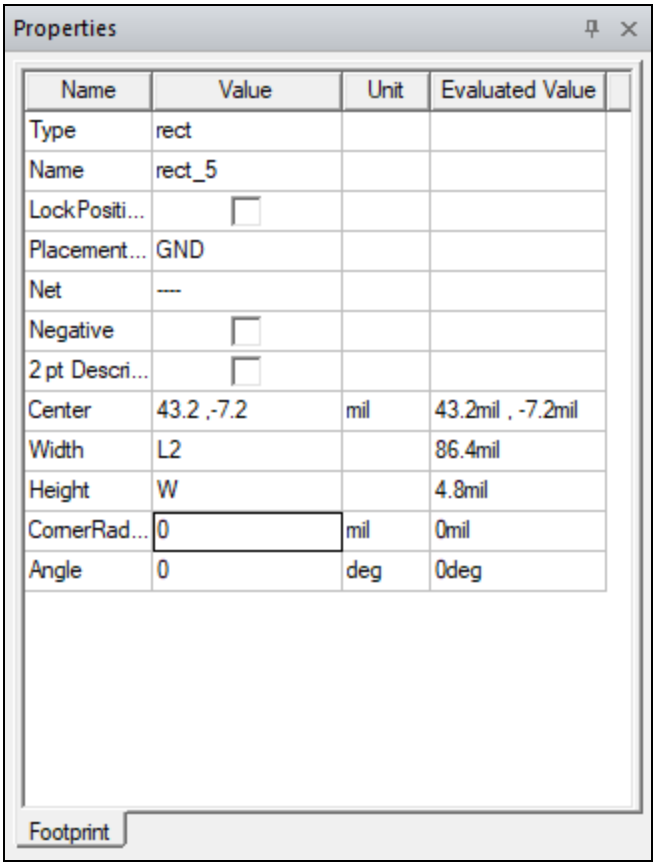
5. Select the new object (i.e., **L2**) to populate the **Properties** window. Then make the following changes:

a. Ensure the **2 pt Description** option is **not** selected.
b. Enter **L2** in the **Width** field. Then press **Enter** or **Tab** to open the **Add Variable** window.

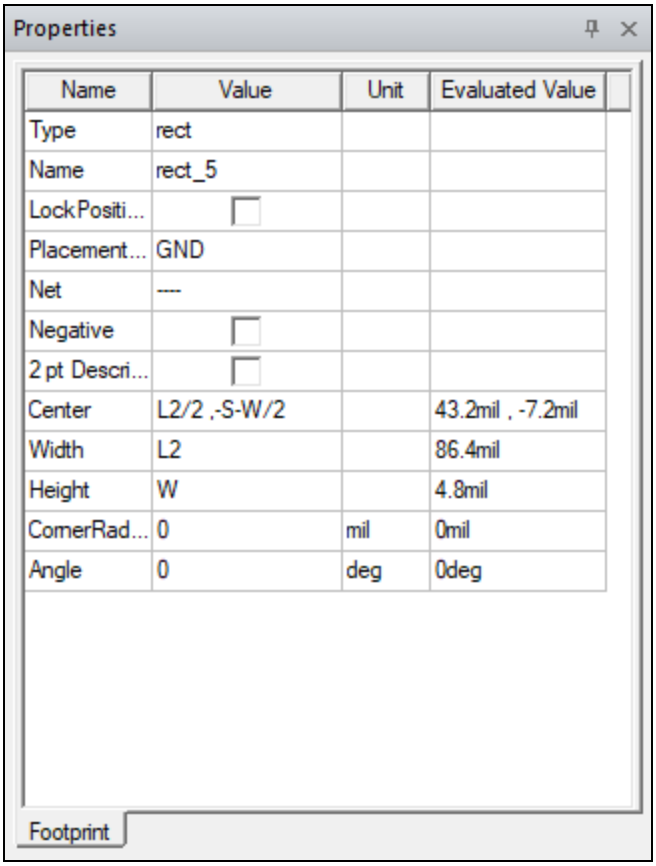


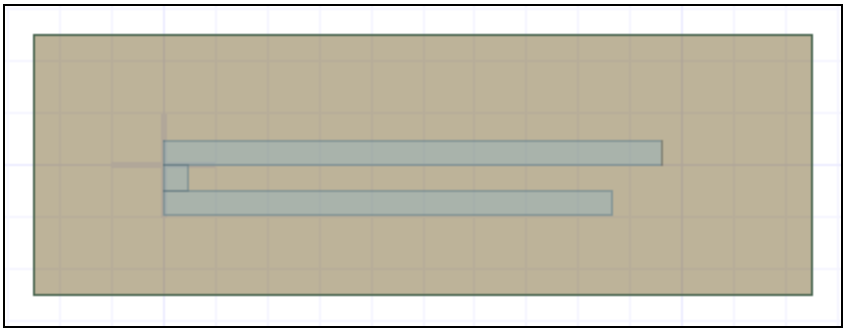
c. Select **mil** from the **Unit** drop-down menu and ensure **86.4** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.

d. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.



e. Enter **L2/2, -S-W/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates.

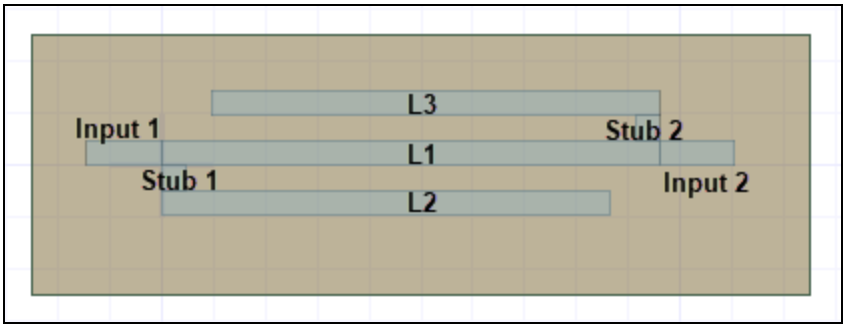




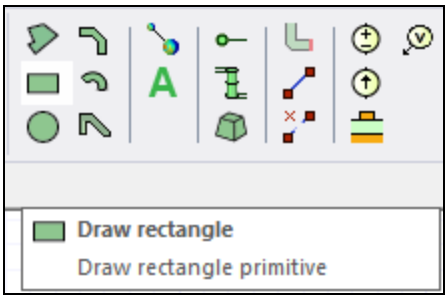
Continue to [Creating Stub 2](#).

Creating Stub 2

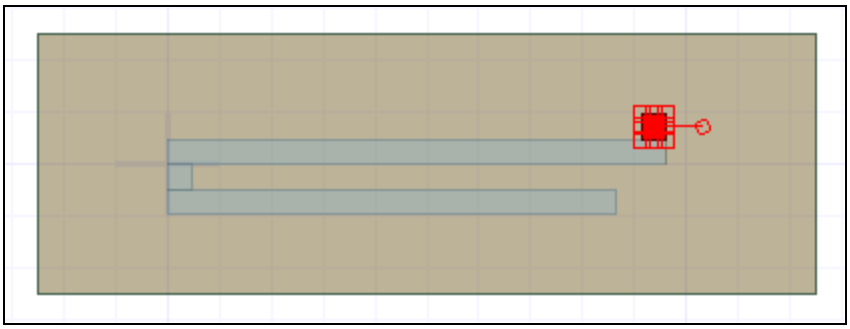
Complete these steps to create the object Stub 2 and then parameterize the object.



1. From the **Layout** tab, click **Draw rectangle**.

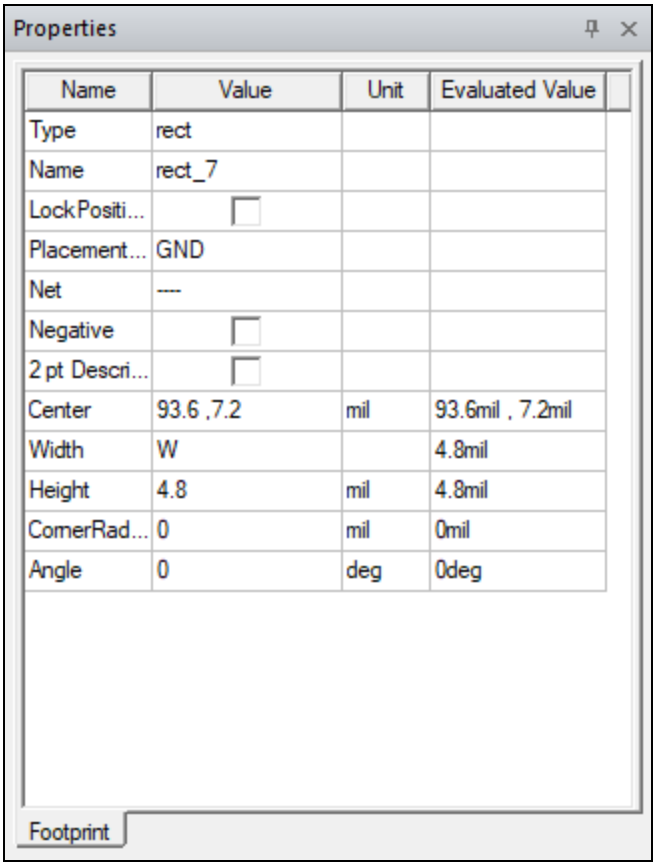


2. Click the upper-right corner of **L1** (i.e., **96, 4.8**).
3. Press **Tab** to move the cursor to the **Delta X** coordinate field at the bottom of the **Layout Editor**. Then enter **-4.8**.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field and press **Enter** to complete the new rectangle.

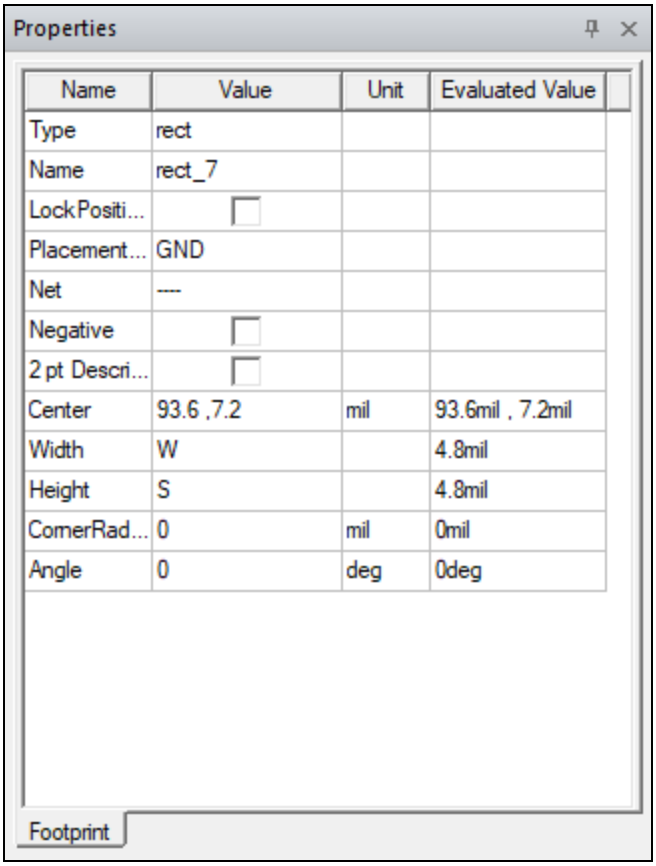


5. Select the new object (i.e., **Stub 2**) to populate the **Properties** window. Then make the following changes:

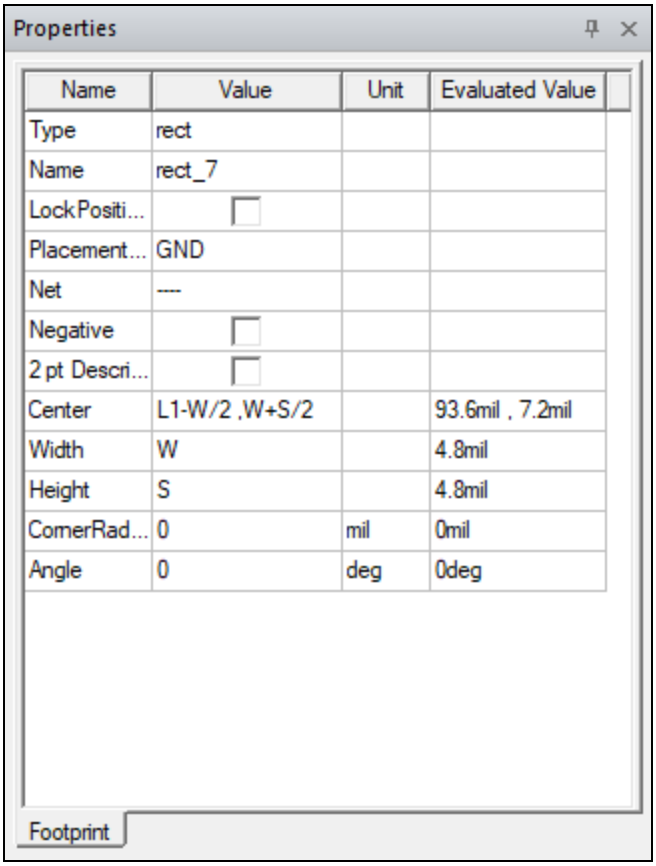
a. Ensure the **2 pt Description** option is **not** selected.
b. Enter **W** in the **Width** field. Then press **Enter** or **Tab**.

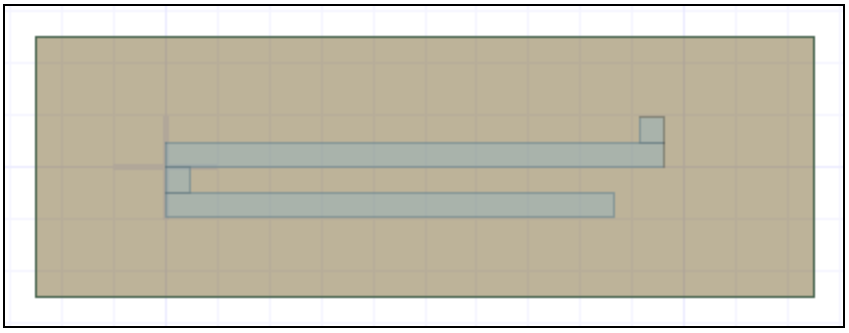


c. Enter **S** in the **Height** field. Then press **Enter** or **Tab**.



d. Enter **L1-W/2 , W+S/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates.

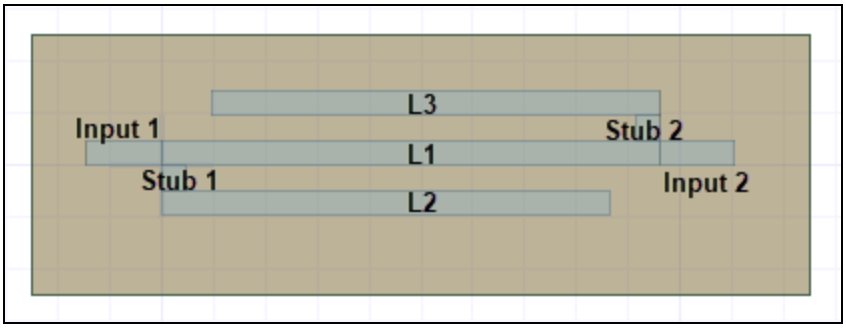




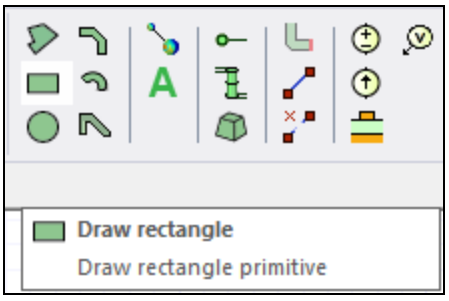
Continue to Creating L3.

Creating L3

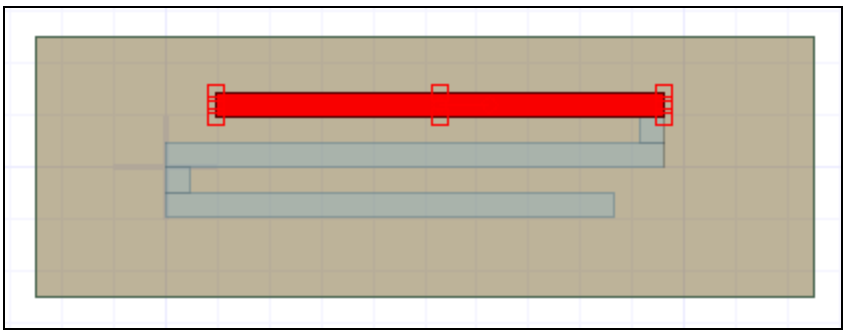
Complete these steps to create the object L3 and then parameterize the object.



1. From the **Layout** tab, click **Draw rectangle**.

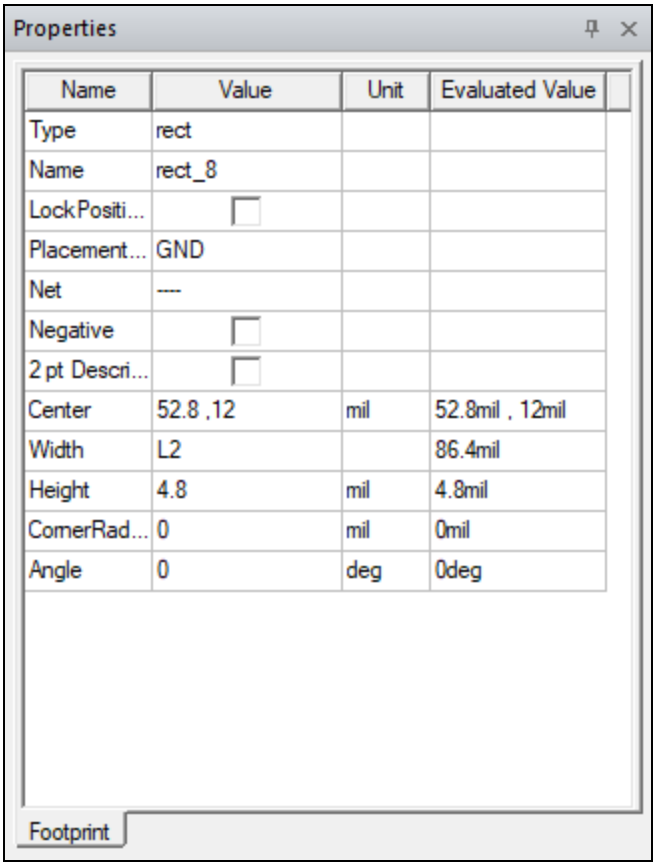


2. Click the upper-right corner of **Stub 2** (i.e., **96, 9.6**).
3. Press **Tab** to move the cursor to the **Delta X** coordinate field at the bottom of the **Layout Editor**. Then enter **-86.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field and press **Enter** to complete the new rectangle.

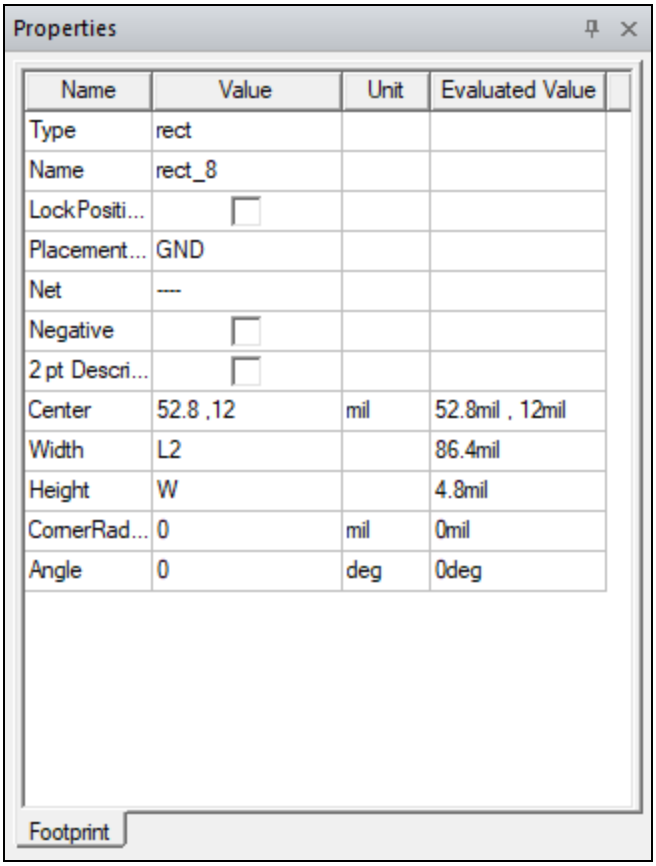


5. Select the new object (i.e., **L3**) to populate the **Properties** window. Then make the following changes:

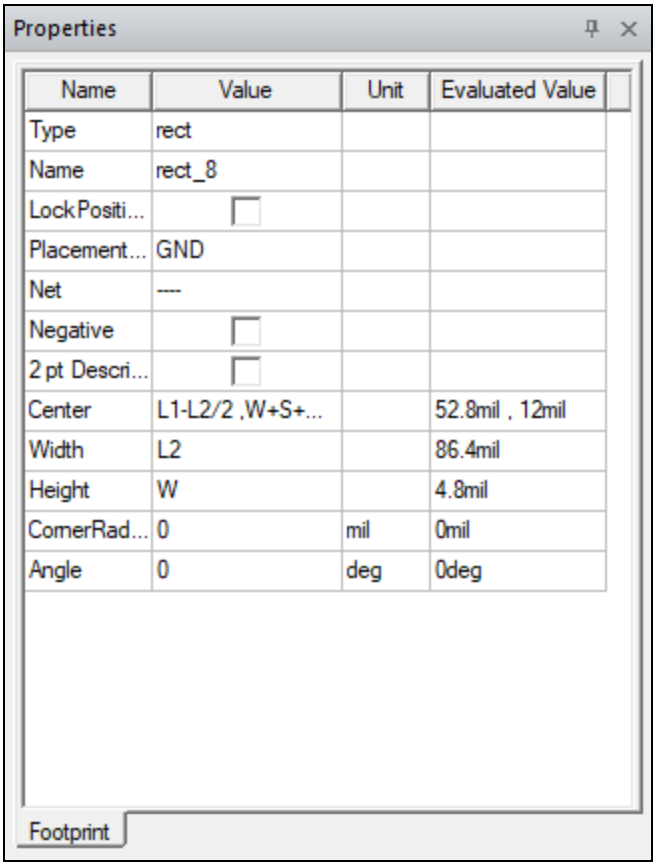
a. Ensure the **2 pt Description** option is **not** selected.
b. Enter **L2** in the **Width** field. Then press **Enter** or **Tab**.

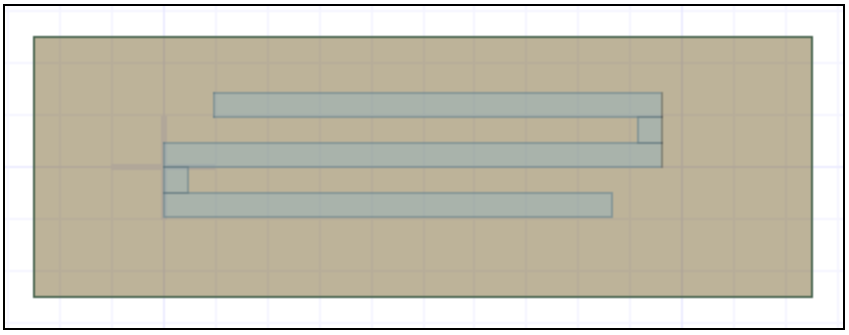


c. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.



d. Enter **L1-L2/2, W+S+W/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates.

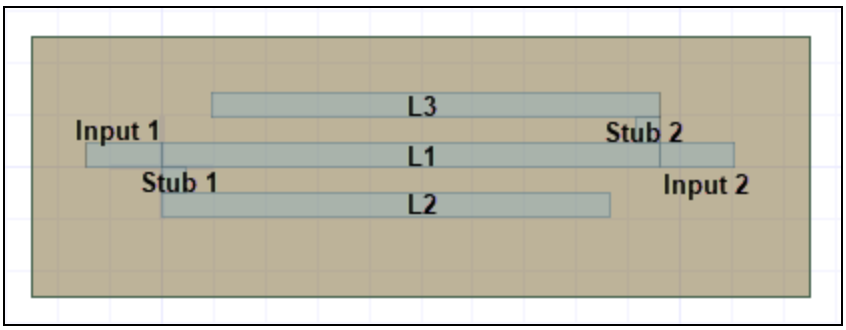




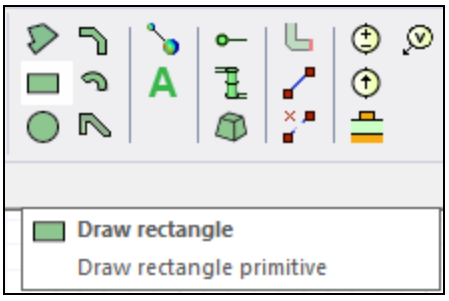
Continue to Creating Input 1.

Creating Input 1

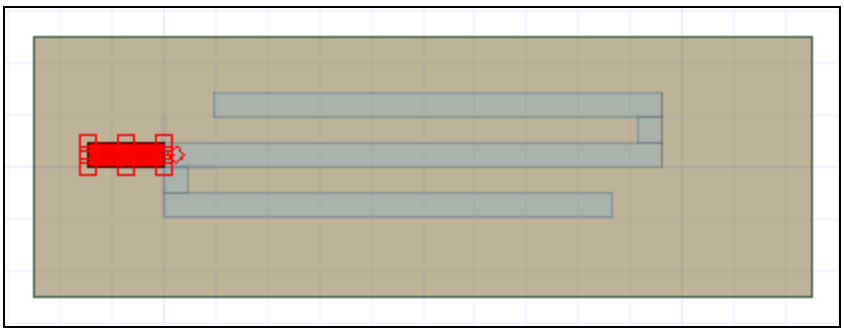
Complete these steps to create the object Input 1 and then parameterize the object.



1. From the **Layout** tab, click **Draw rectangle**.

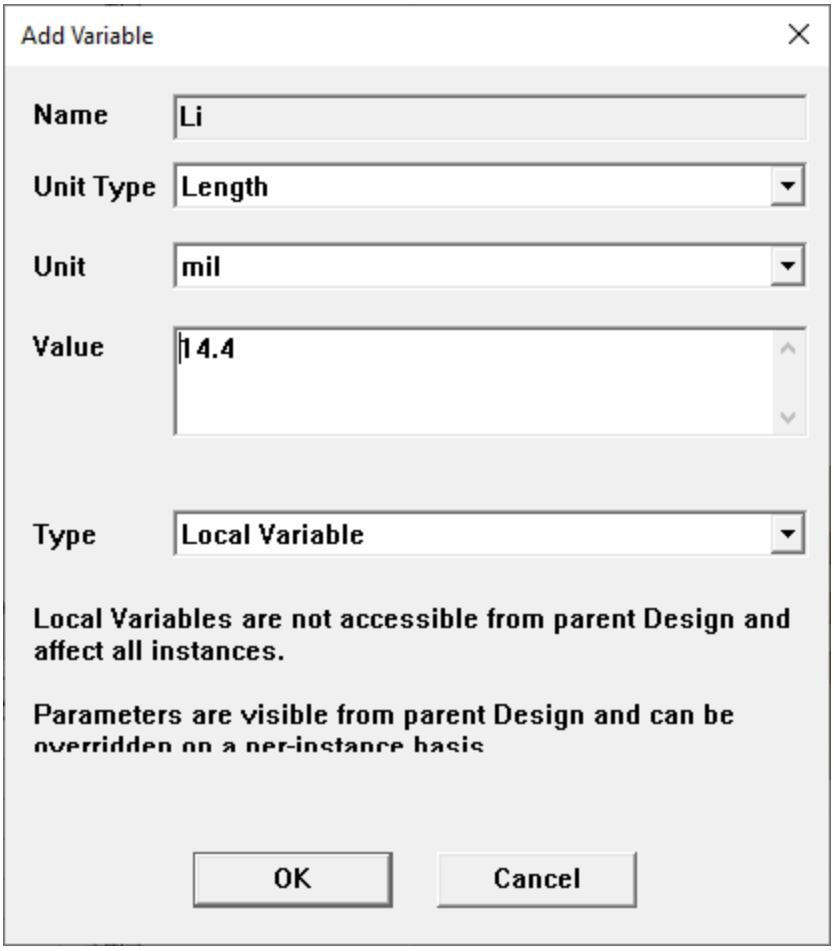


2. Click the lower-left corner of **L2** (i.e., **0, 0**).
3. Press **Tab** to move the cursor to the **Delta X** coordinate field at the bottom of the **Layout Editor**. Then enter **-14.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field and press **Enter** to complete the new rectangle.



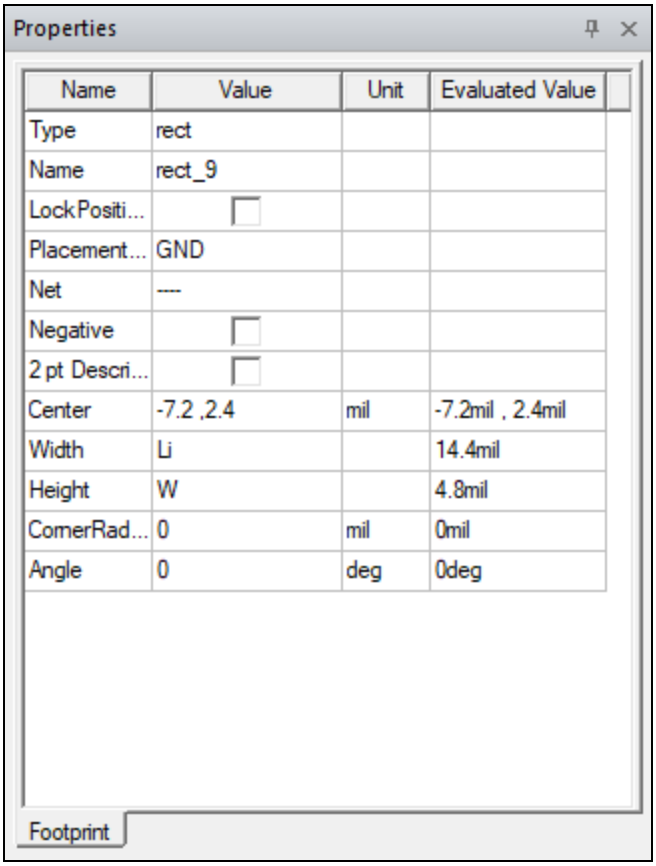
5. Select the new object (i.e., **Input 1**) to populate the **Properties** window. Then make the following changes:
    a. Ensure the **2 pt Description** option is **not** selected.

b. Enter **Li** in the **Width** field. Then press **Enter** or **Tab** to open the **Add Variable** window.

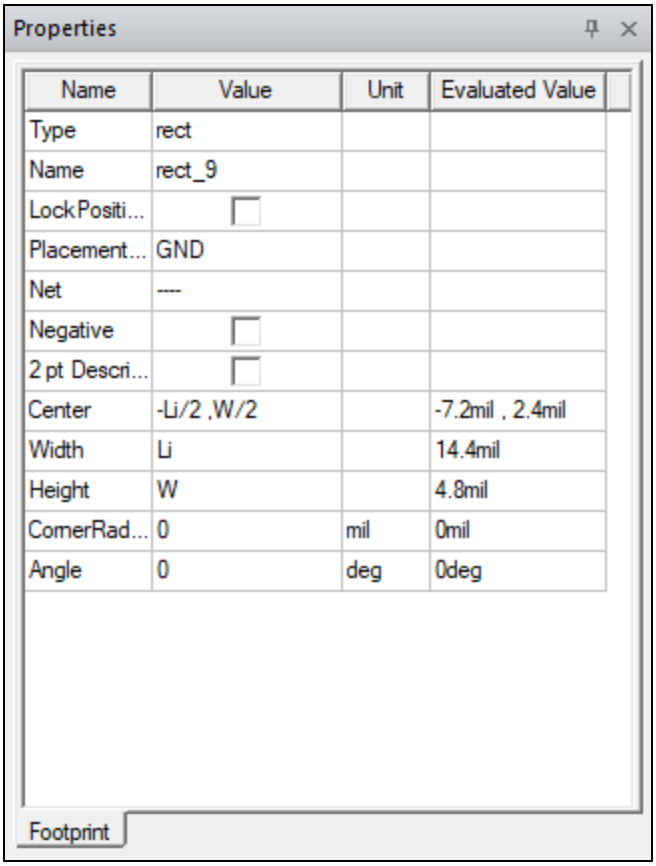


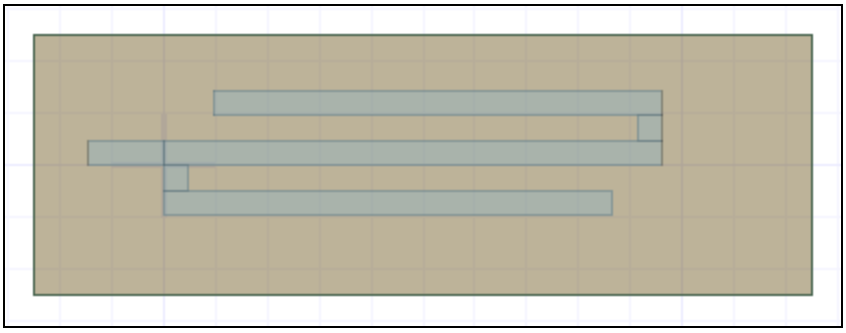
c. Select **mil** from the **Unit** drop-down menu and ensure **14.4** is entered in the **Value** field. Then click **OK** to close the **Add Variable** window.

d. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.



e. Enter **-Li/2, W/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates.

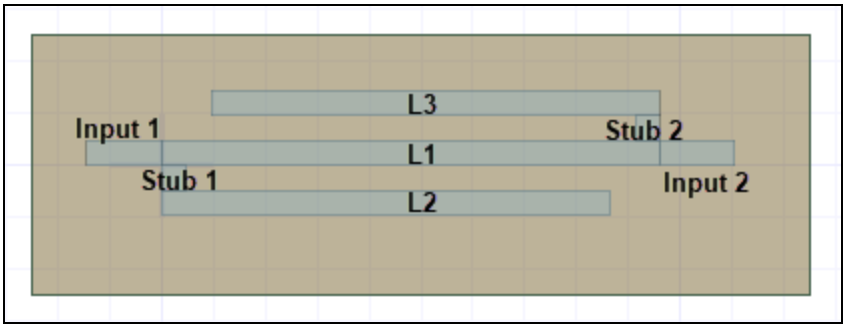




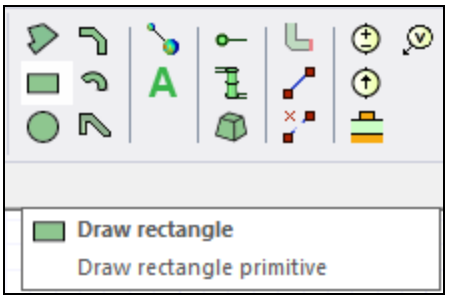
Continue to [Creating Input 2](#).

Creating Input 2

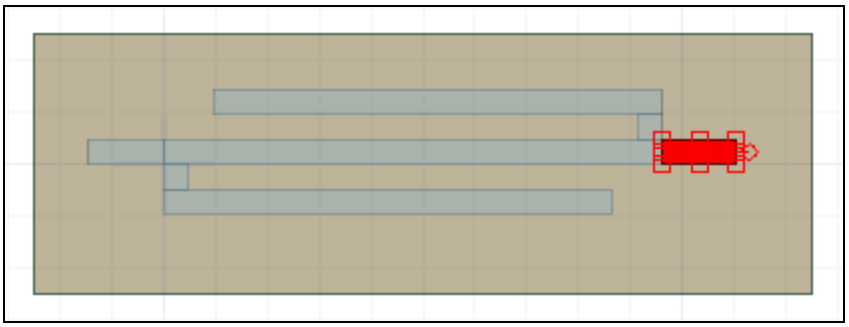
Complete these steps to create the object Input 2 and then parameterize the object.



1. From the **Layout** tab, click **Draw rectangle**.

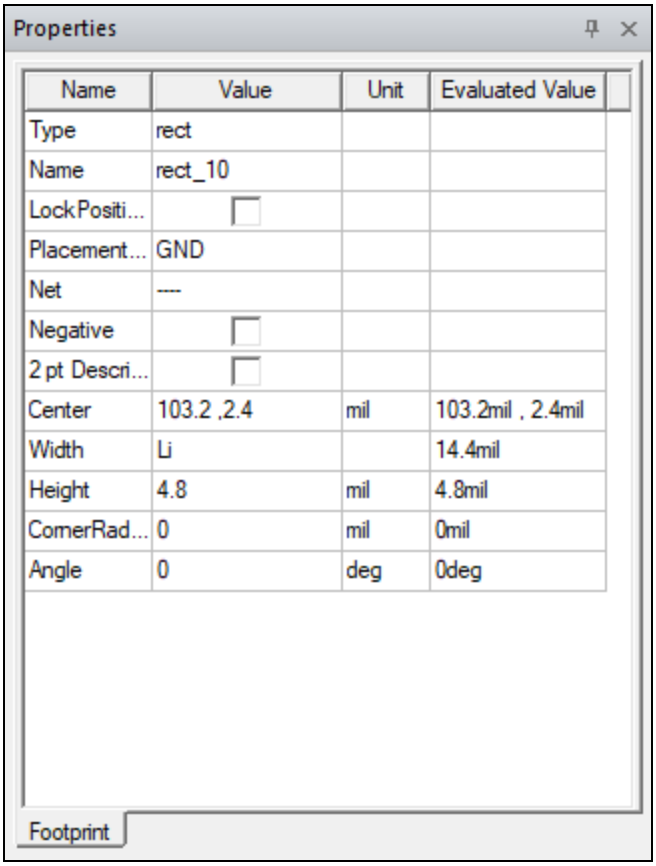


2. Click the lower-right corner of **L1** (i.e., **96, 0**).
3. Press **Tab** to move the cursor to the **Delta X** coordinate field at the bottom of the **Layout Editor**. Then enter **14.4** in the field.
4. Press **Tab** to move the cursor to the **Delta Y** coordinate field. Then type **4.8** in the field and press **Enter** to complete the new rectangle.

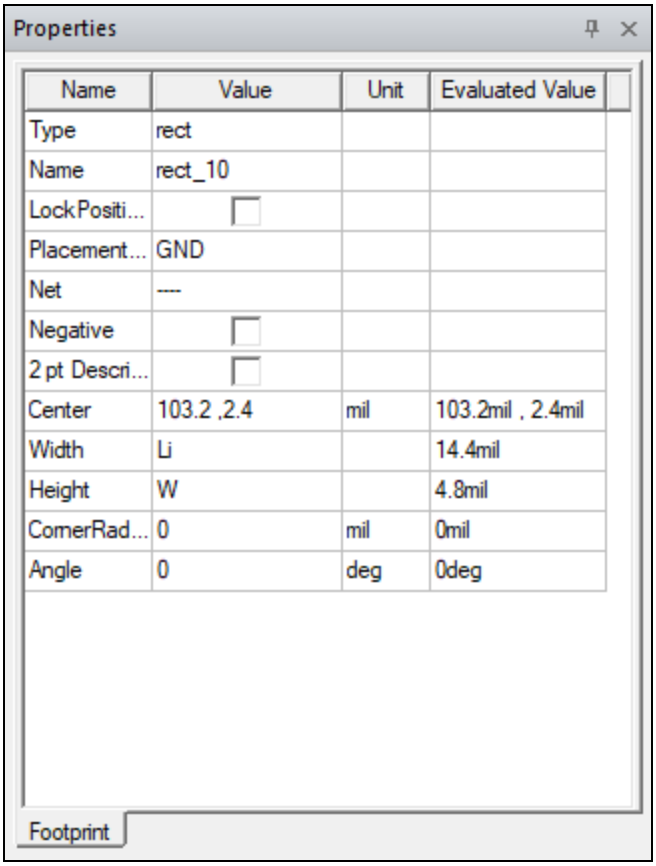


5. Select the new object (i.e., **Input 2**) to populate the **Properties** window. Then make the following changes:

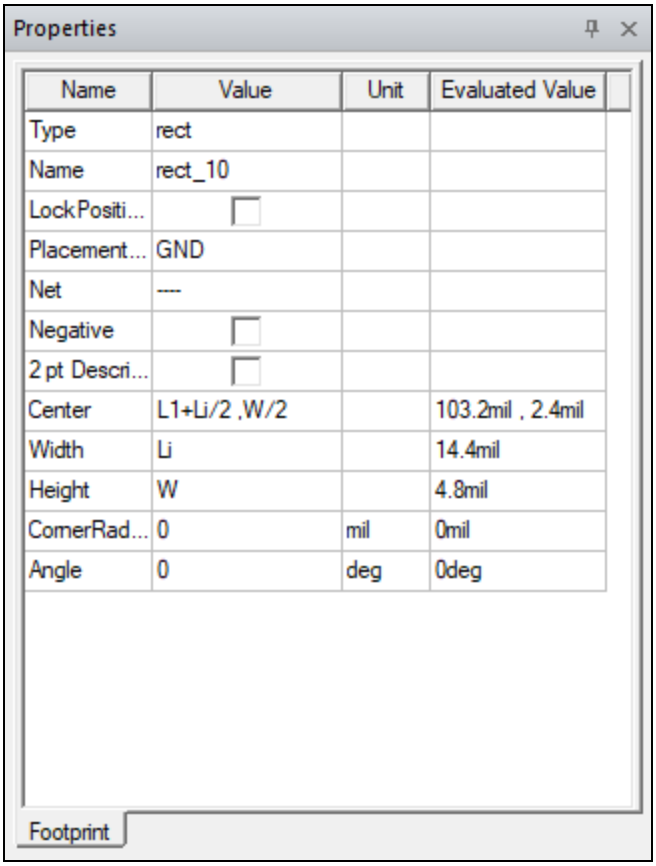
a. Ensure the **2 pt Description** option is **not** selected.
b. Enter **Li** in the **Width** field. Then press **Enter** or **Tab**.

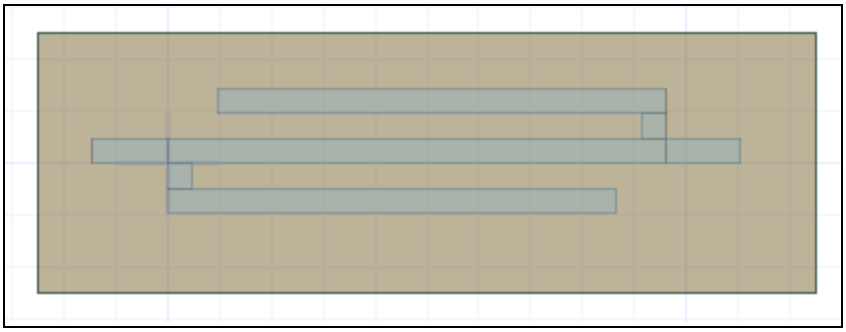


c. Enter **W** in the **Height** field. Then press **Enter** or **Tab**.



d. Enter **L1+Li/2, W/2** in the **Center** field and press **Enter** or **Tab** to accept the coordinates.





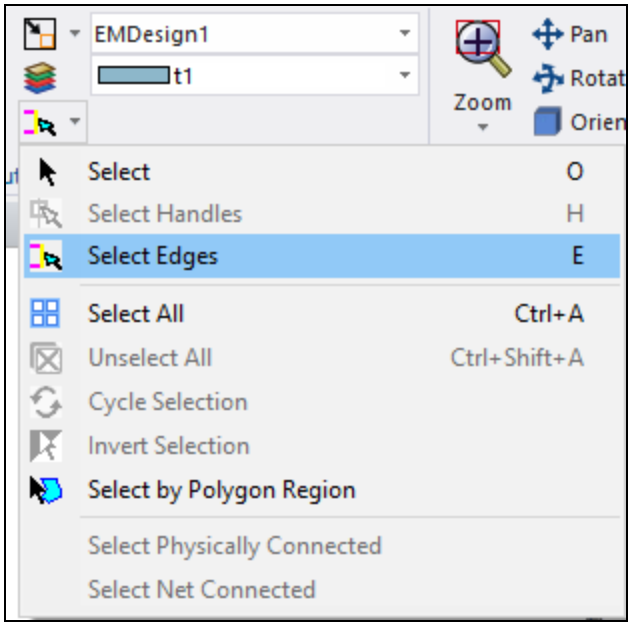
Continue to Creating Edge Ports.

Creating Edge Ports

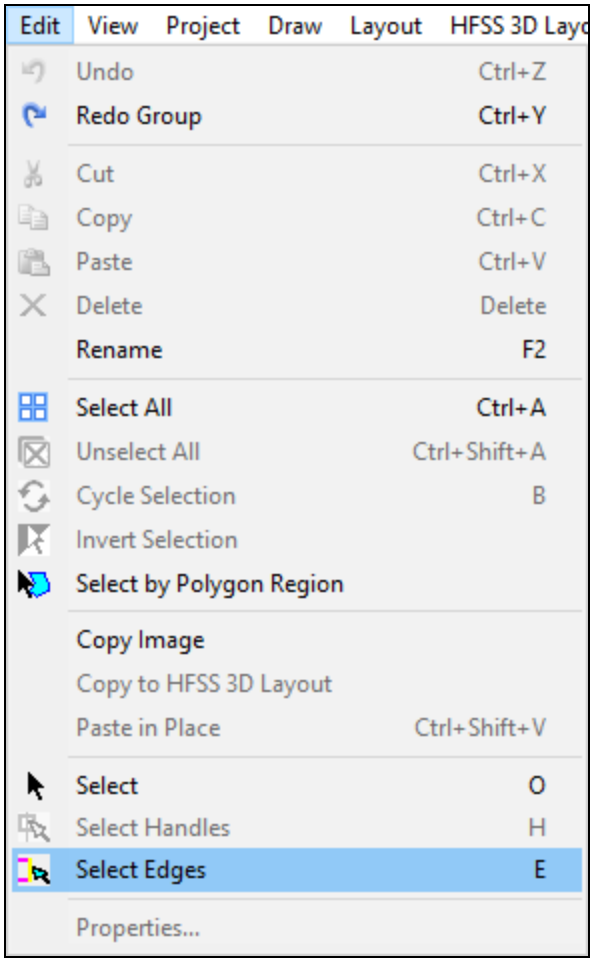
Complete these steps to add two edge ports to the model.

1. To create the first port (i.e., ***Port1***), first do any of the following to switch to edge selection mode:

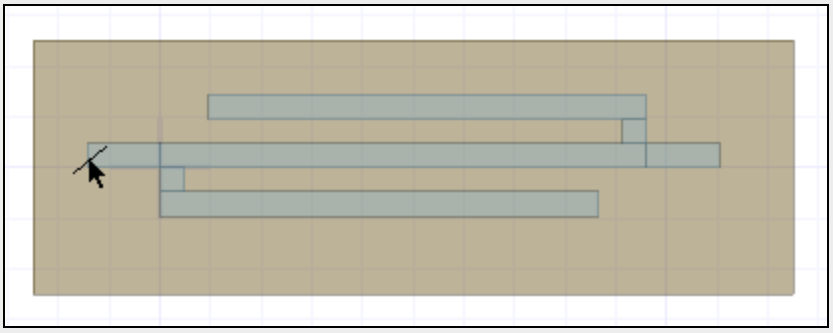
- Press **E** to immediately enter **Select Edges** mode.
- From the **Layout** tab, choose **Select Edges** from the cursor drop-down menu.



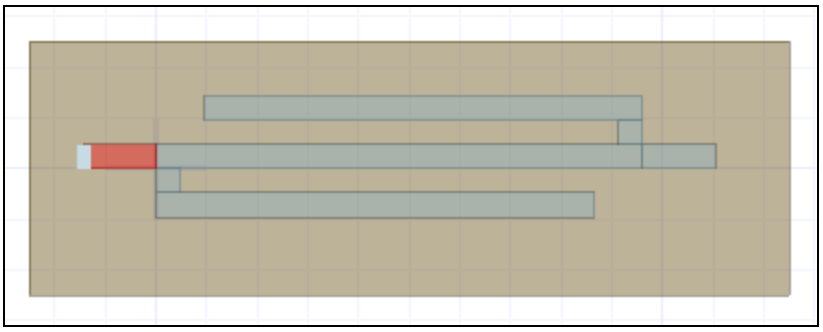
- From **Edit**, click **Select Edges**.



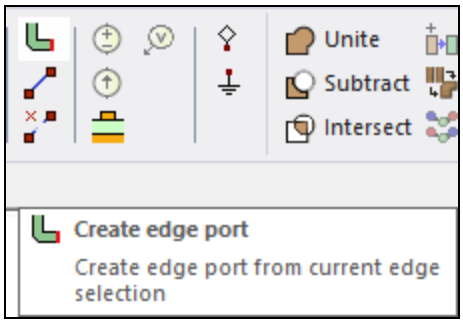
Note: Once **Select Edges** is chosen, the cursor changes: a diagonal line crosses the tip of the arrow.



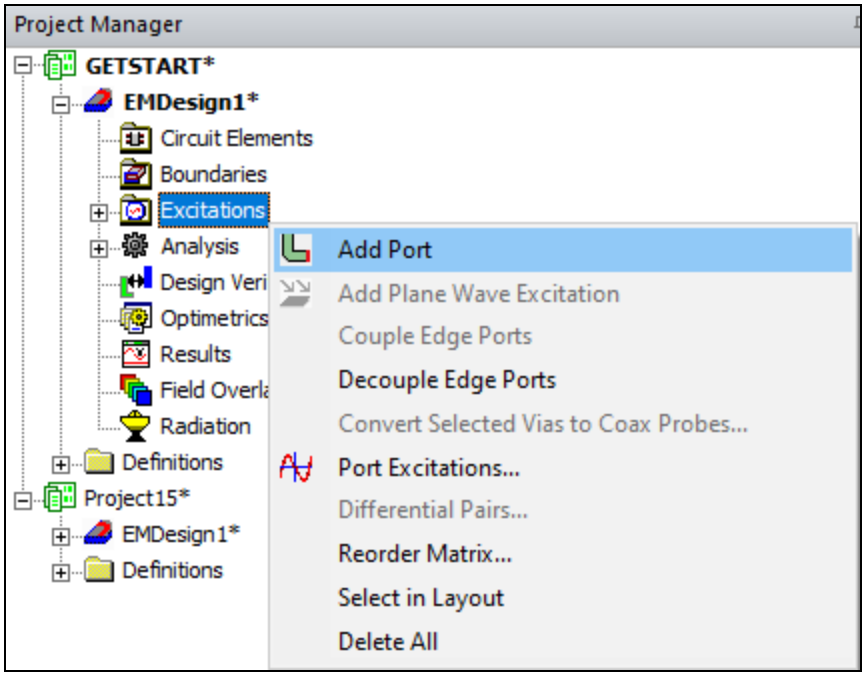
2. Click the left edge of **Input 1** to select it.



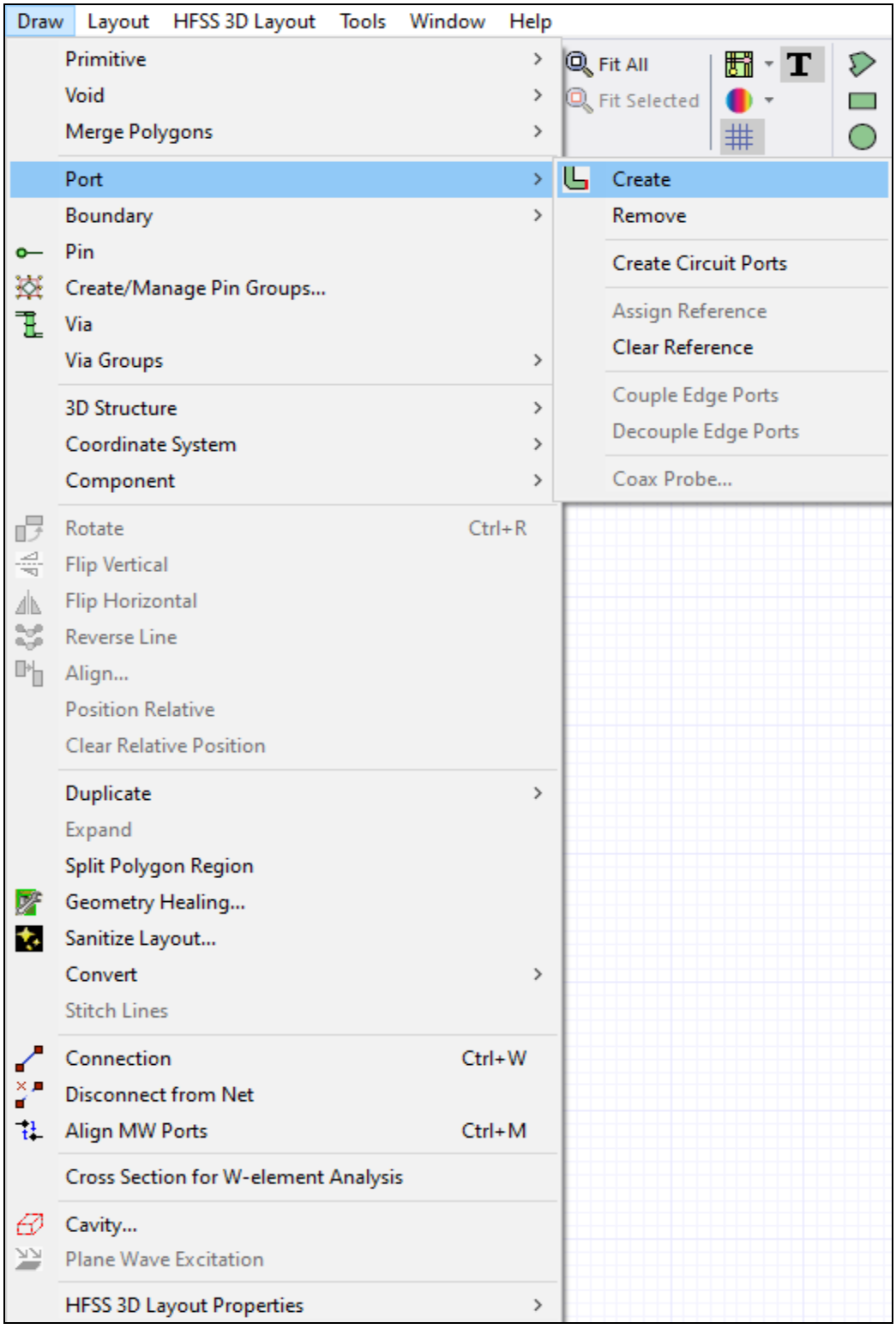
3. To add the first port to the leftmost rectangle, do one of the following:
   - From the **Layout** tab, select **Create edge port**.



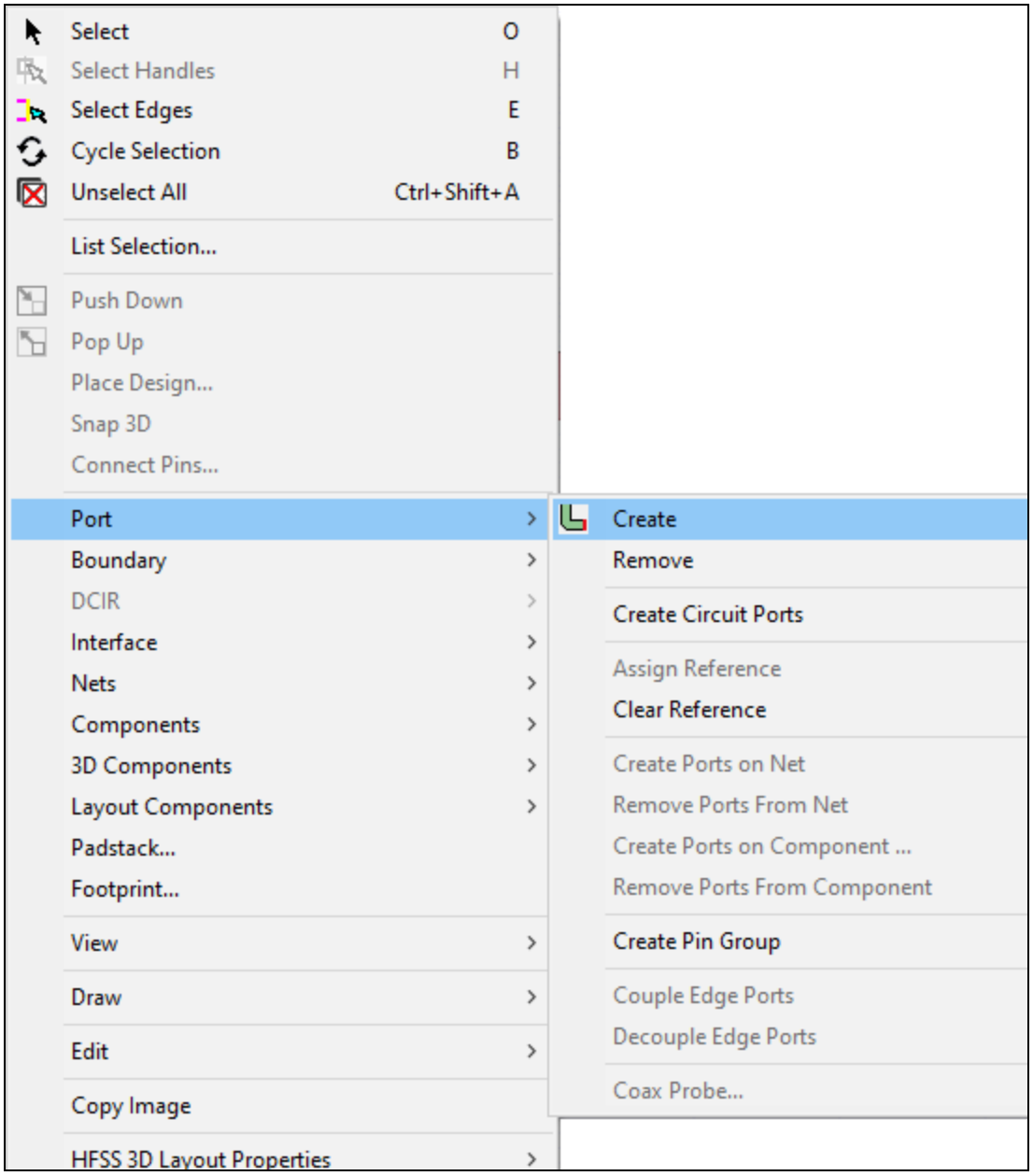
- From the **Project Manager** window, expand the **Project Tree** and **[*active design folder*]**. Then right-click **Excitations** and select **Add Port**.



- From **Draw**, select **Port** > **Create**.

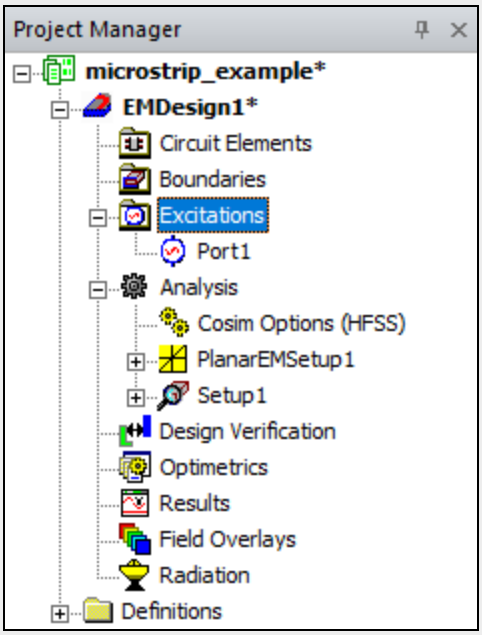


- Right-click in the **Layout Editor** and select **Port** > **Create**.

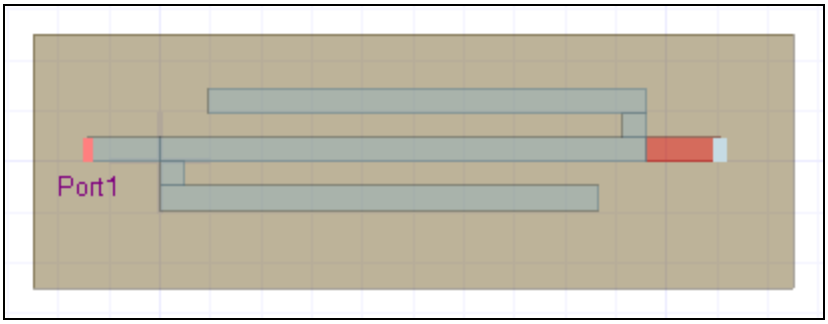


Note:

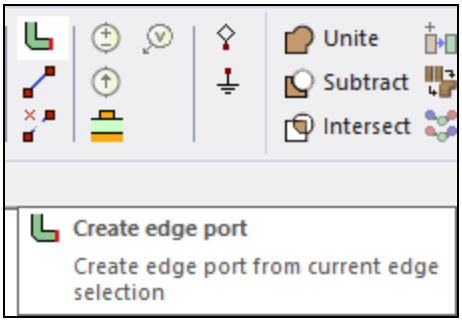
Once a port is created, it appears in the **Project Manager** > **Project Tree** > **[*active design folder*]** > **Excitations** folder.

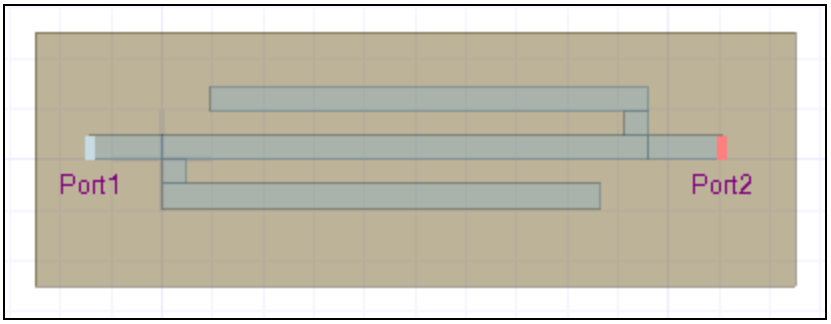


4. Create the second port (i.e., ***Port2***) by doing the following:
   a. If appropriate, press **E** to re-enter **Select Edges** mode.
   b. Click the right edge of **Input 2** to select it.



c. From the **Layout** tab, select **Create edge port**.





Note:

In an earlier **Getting Started Guide**, the individual objects comprising the trace layer were united into a single object. However, because this model is parametric, do **not** unite the objects. Unlike the 3D Modeler used for conventional HFSS designs, the **Layout Editor** does not maintain the full parametric history of the model construction. If the trace objects are united, the parameters defined for the individual rectangles no longer have any effect from the geometry. Once the individual rectangles are merged into a complex polygon, the vertices of that polygon are defined using their absolute numerical coordinates. Therefore, altering the design variables no longer has an effect on the model because the variables are no longer used to define the united object. All of the edges of the conducting objects in this exercise are coincident with their adjacent objects' edges. Whether united or not, the trace objects behave like a single conducting part and the solver treats the objects as a contiguous conducting part.

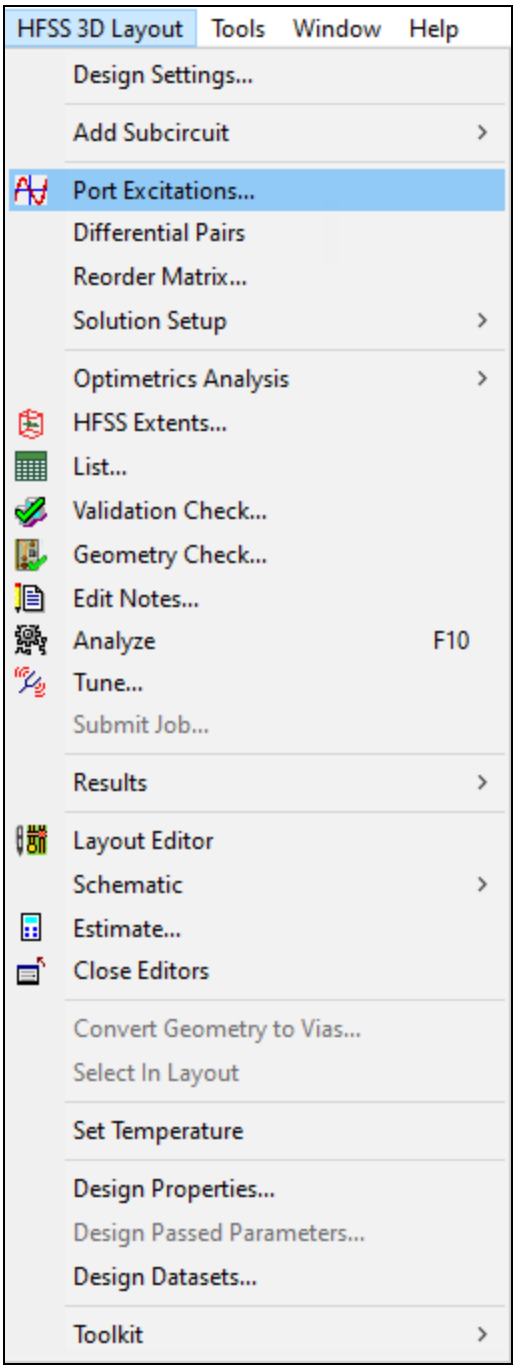
Continue to Setting Port Excitations.

Setting Port Excitations

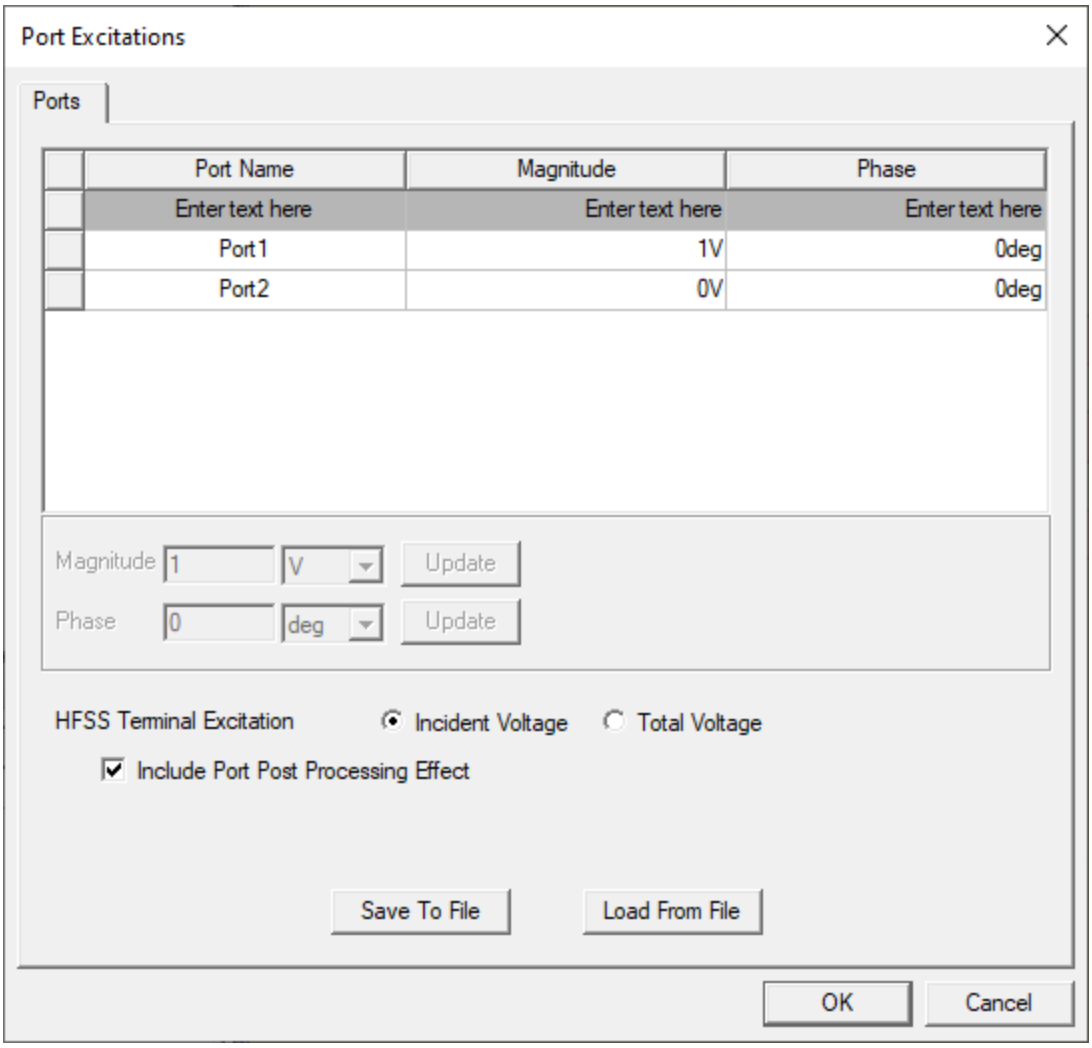
In order to create meaningful results with the microstrip filter design, one edge port should act as the filter input and the other as the filter output. By default, a 1 volt excitation at 0 degrees phase

is applied to each edge port (i.e., **Port1** and **Port2**). Complete these steps to modify the port excitations to specify zero volts at **Port2**, allowing it to act as the filter output.

1. From **HFSS 3D Layout**, select **Port Excitations** to open the **Port Excitations** window.



2. From the **Port Excitations** window, enter **0V** in the **Port2 Magnitude** field.



3. Click **OK** to close the **Port Excitations** window and return to the **Layout Editor**.
4. **Save** the project, either by navigating to **File** > **Save** or clicking the **Save** button on any of the ribbons.

Continue to [Creating an HFSS Analysis Setup](#).

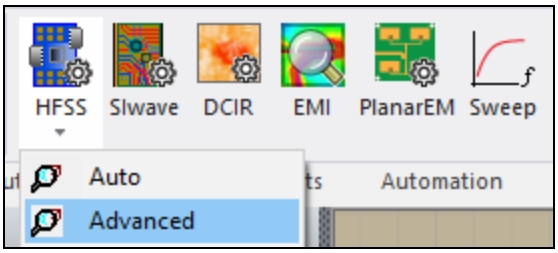
2 - Analysis and Post-Processing

This chapter contains the following topics:

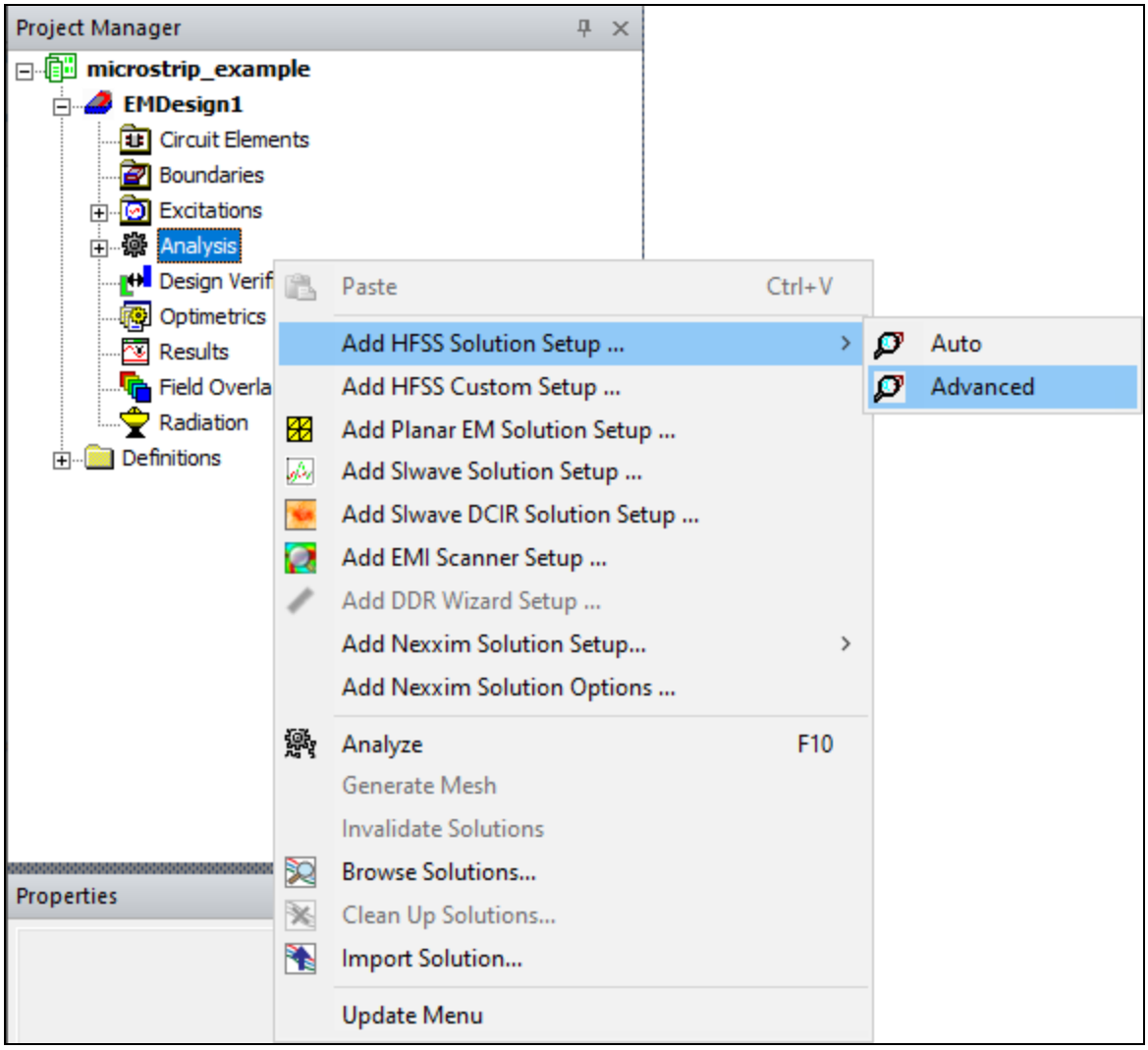
Creating an HFSS Analysis Setup

Solution Setups are listed in the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Analysis**). Complete these steps to add an **HFSS Analysis** setup solution to this project using basic, initial meshing tools.

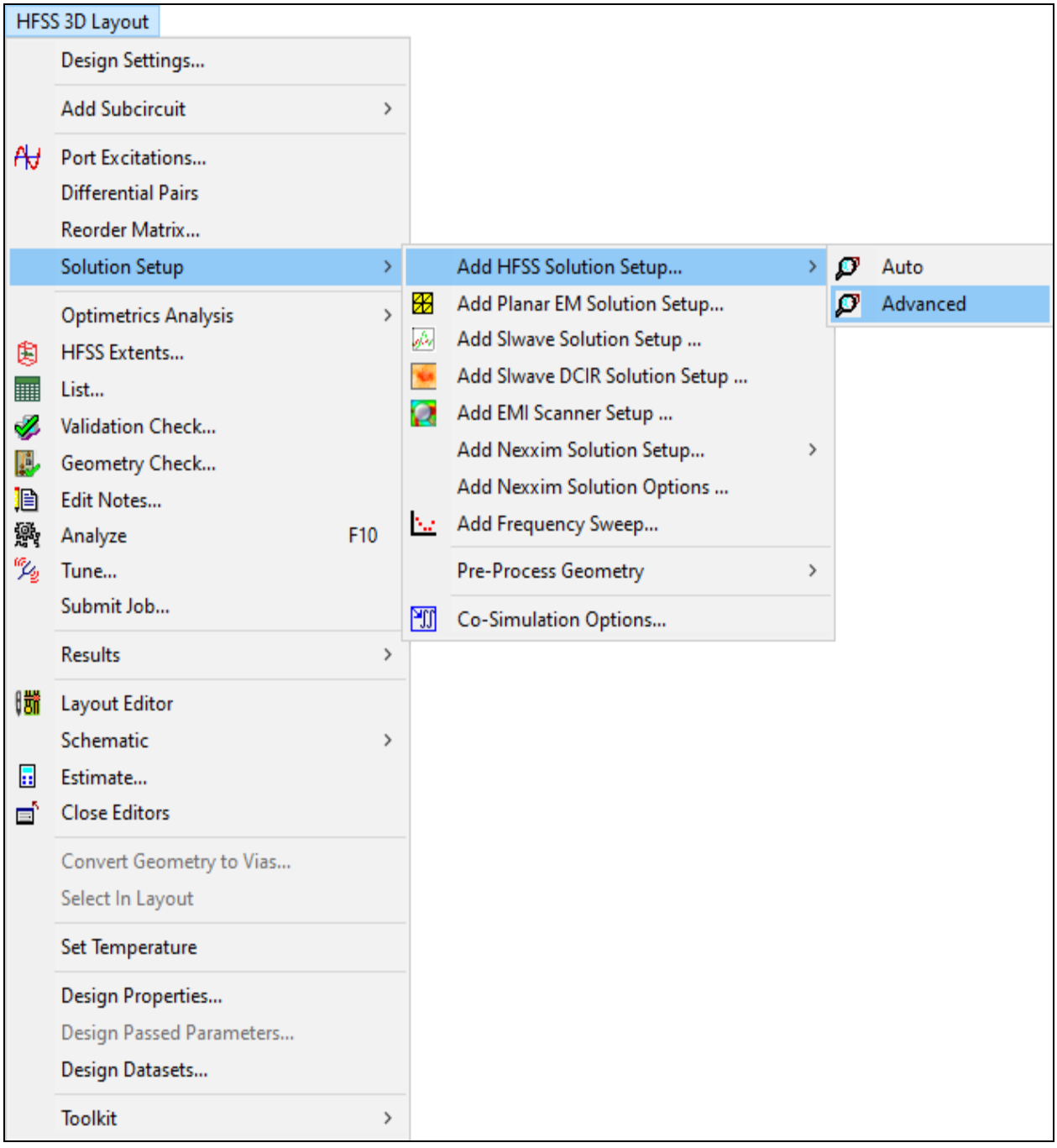
1. Open the **Setup** window by doing one of the following:
   - From the **Simulation** ribbon tab, click **HFSS** > **Advanced**.



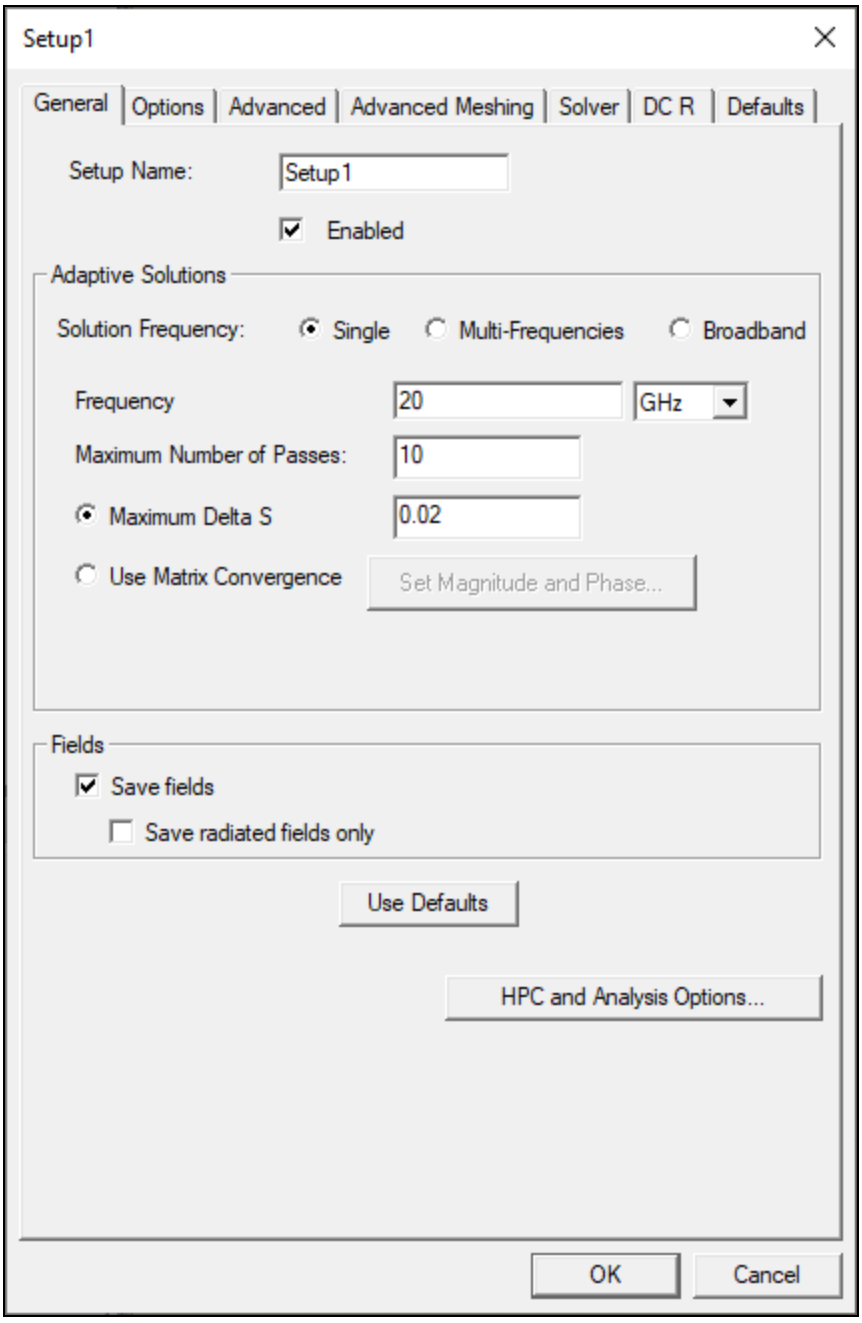
- Right-click **Analysis** in the **Project Manager** window and select **Add HFSS Solution Setup** > **Advanced**.



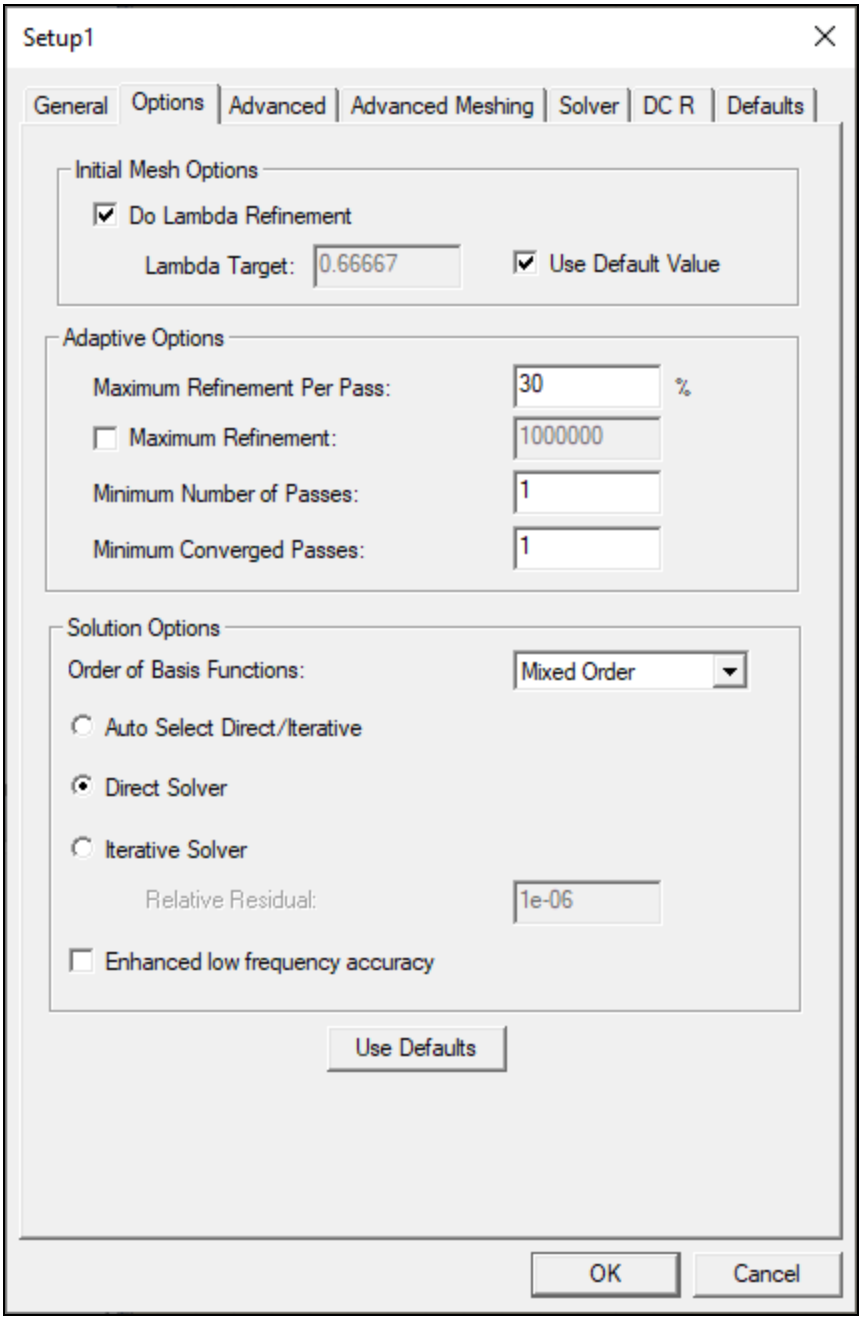
- From **HFSS 3D Layout**, select **Solution Setup** > **Add HFSS Solution Setup** > **Advanced**.



2. Enter **20** in the **Frequency** field.
3. From the **Fields** area, check the **Save fields** box .

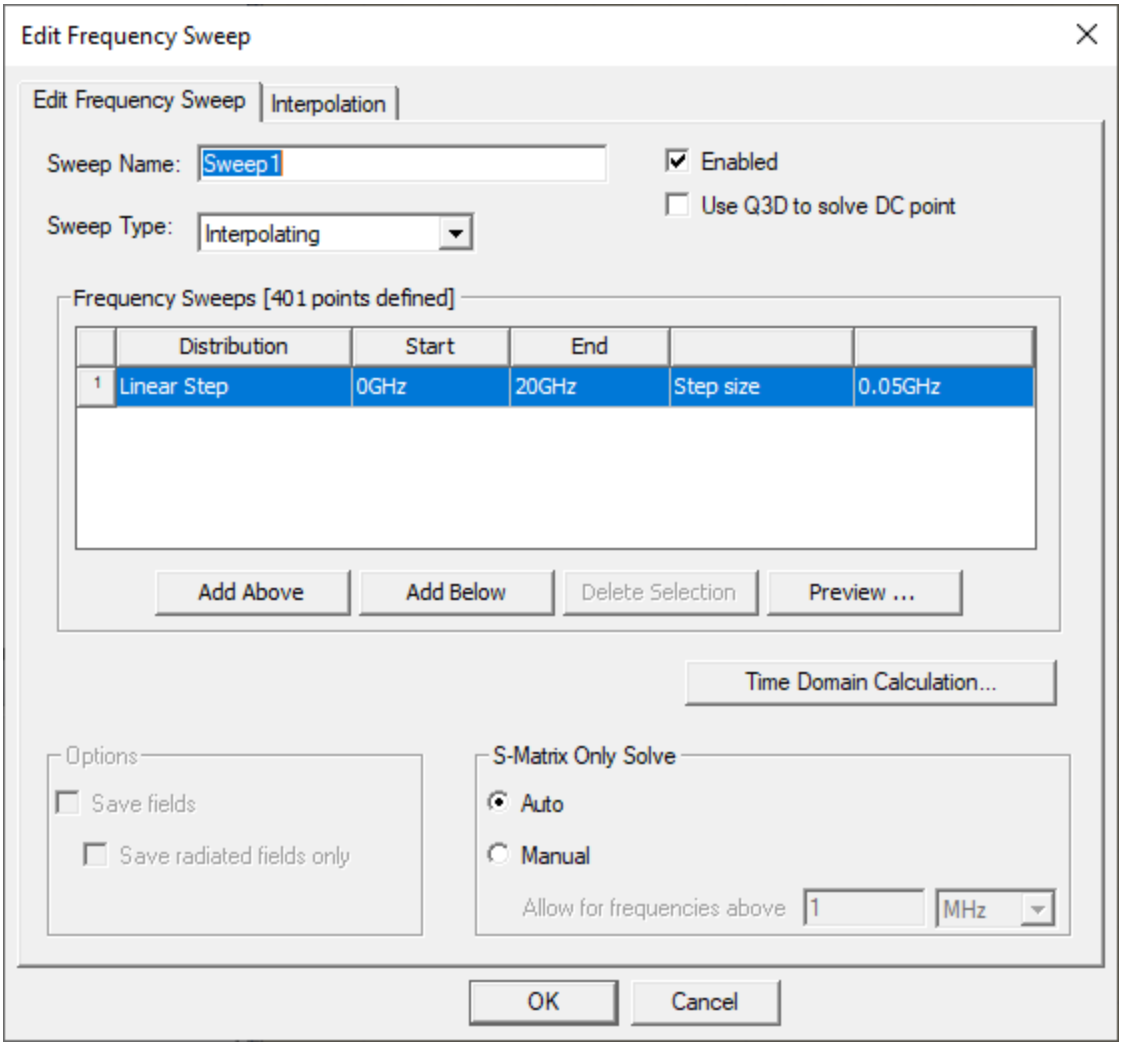


4. Navigate to the **Options** tab.

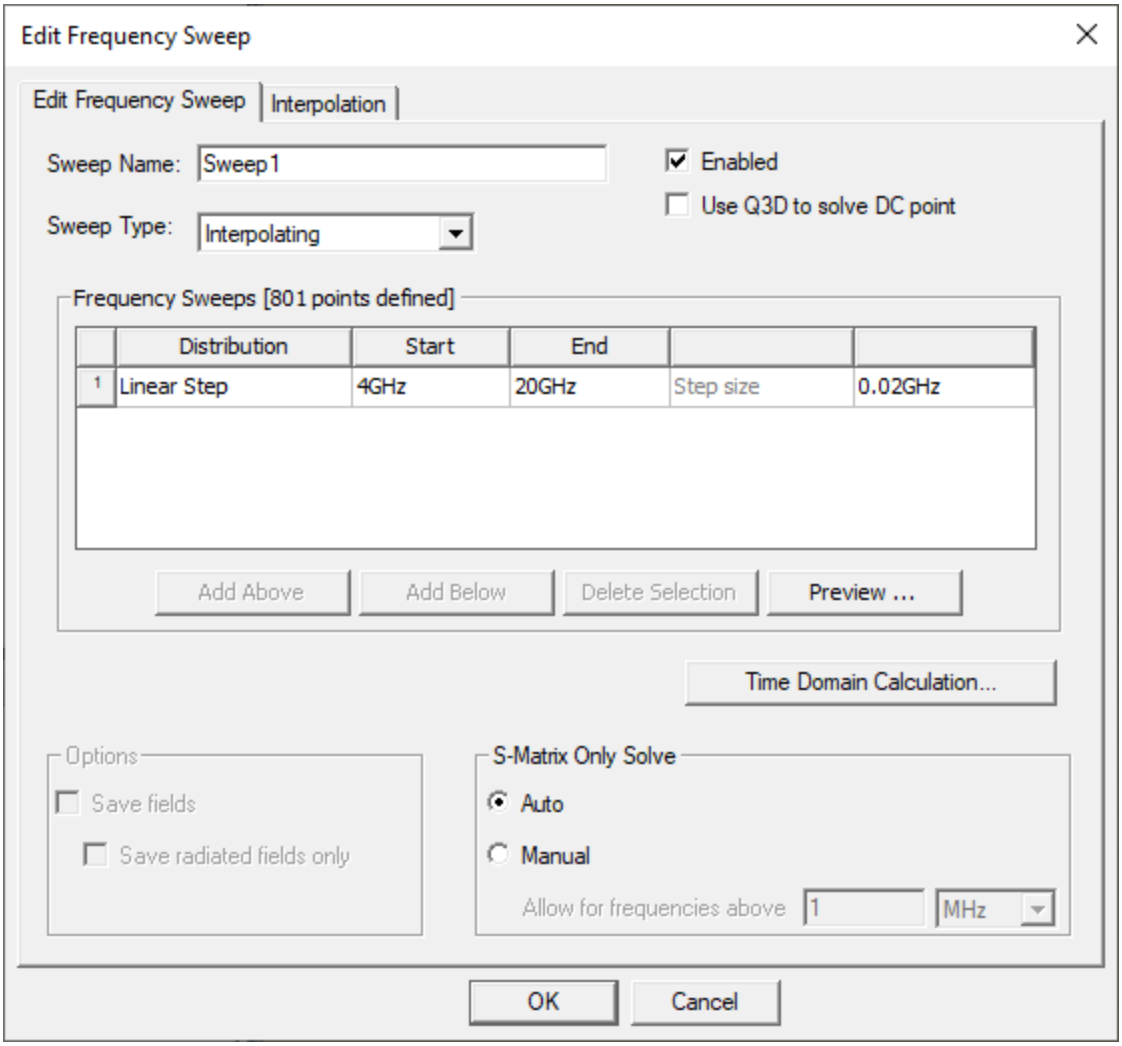


5. Ensure **Mixed Order** is selected from the **Order of Basis Functions** drop-down menu.

6. Click **OK** to close the HFSS Setup window and open the **Edit Frequency Sweep** window.



7. Ensure **Interpolating** is selected from the **Sweep Type** drop-down menu.
8. Ensure **Linear Step** is selected from the **Distribution** drop-down menu.
9. Enter the following parameters in the first row of the **Frequency Sweeps** table:
    - **4** (GHz) in the **Start** field.
    - **20** (GHz) in the **End** field.
    - **0.02** (GHz) in the **Step size** field.



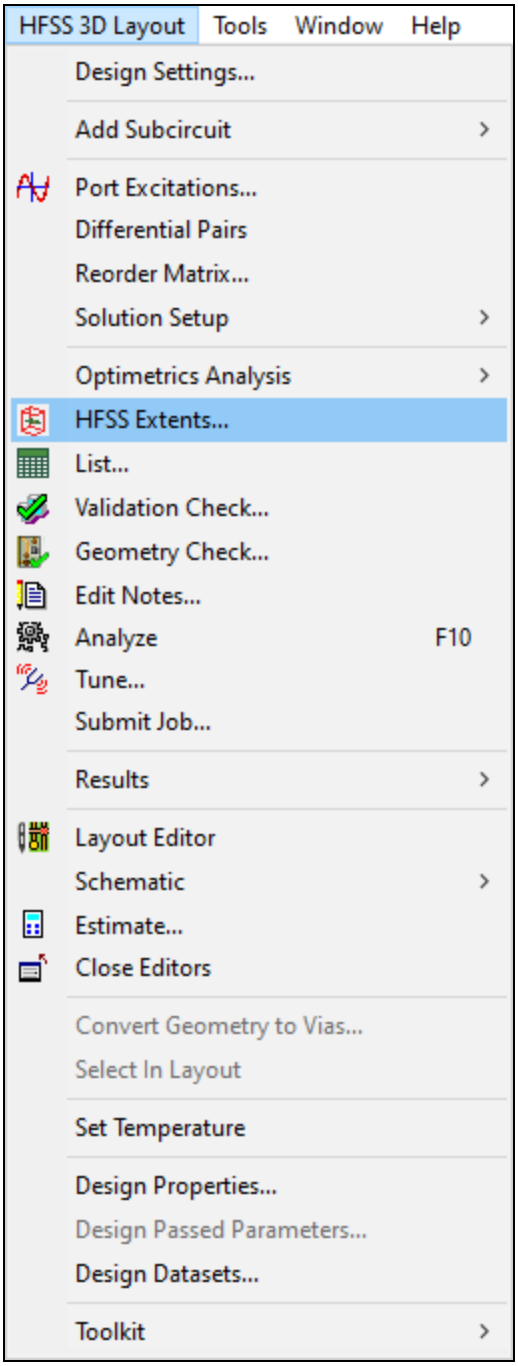
10. Click **OK** to add the interpolating sweep and close the **Edit Frequency Sweep** window.

Continue to [Displaying the HFSS Bounding Box](#).

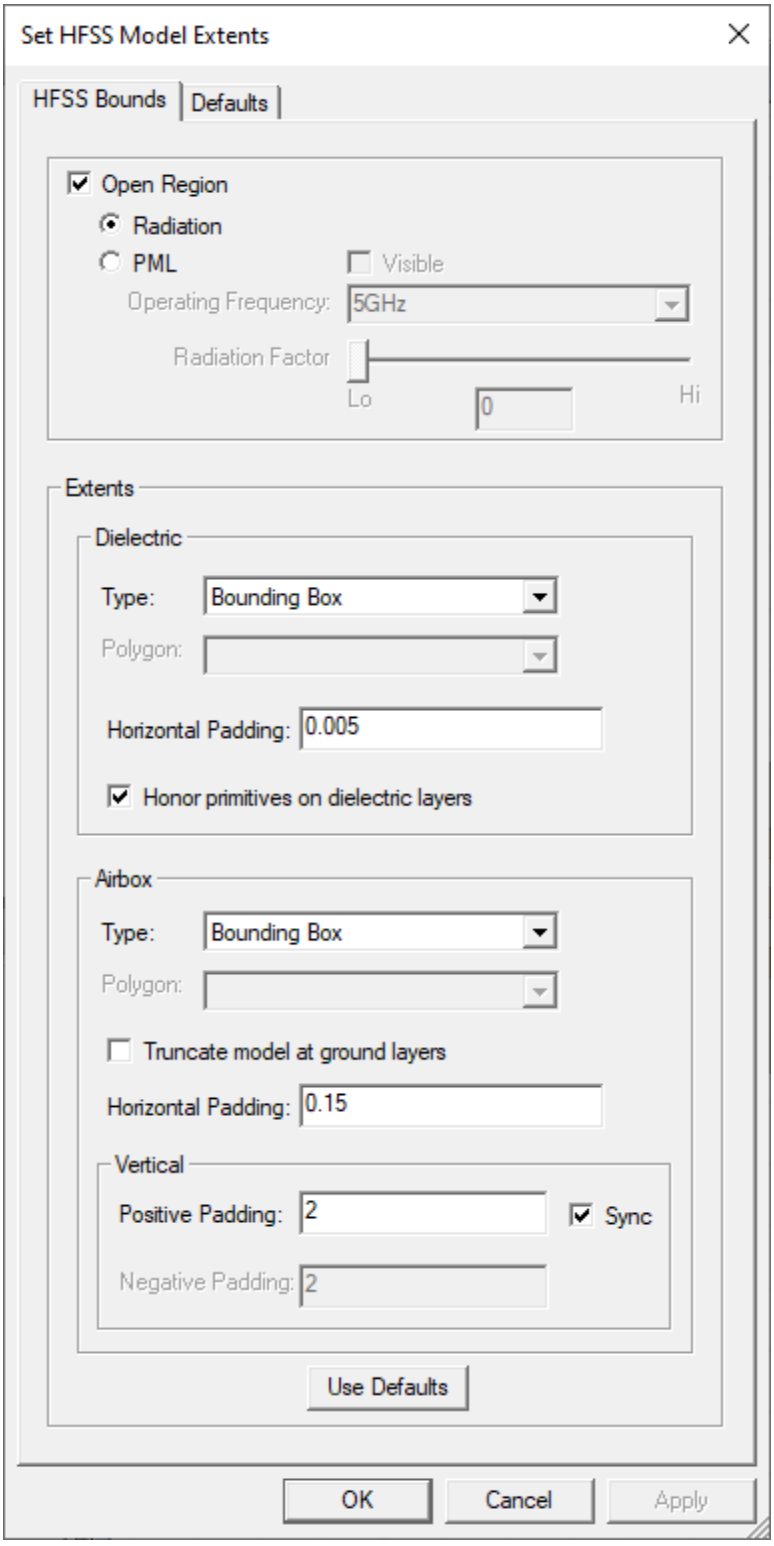
Displaying the HFSS Bounding Box

Complete these steps to display and then hide a bounding box around the complete design in the **Layout Editor**.

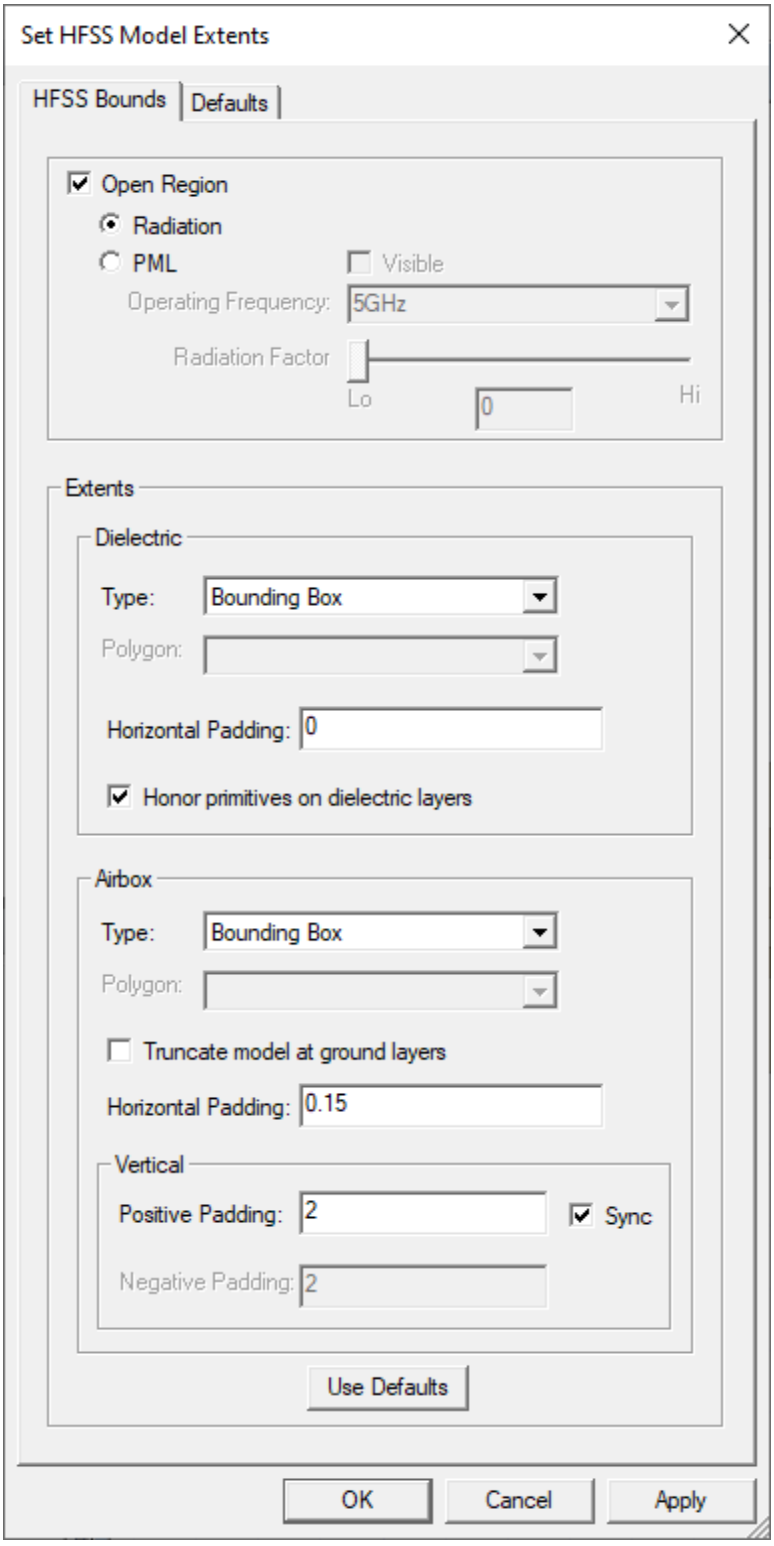
1. From **HFSS 3D Layout**, select **HFSS Extents** to open the **Set HFSS Model Extents** window.



2. Ensure the **Open Region** box is checked and **Radiation** is selected.

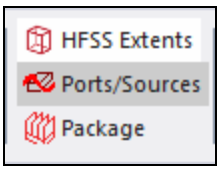


3. From the **Dialectric** area, enter **0** in the **Horizontal Padding** field.

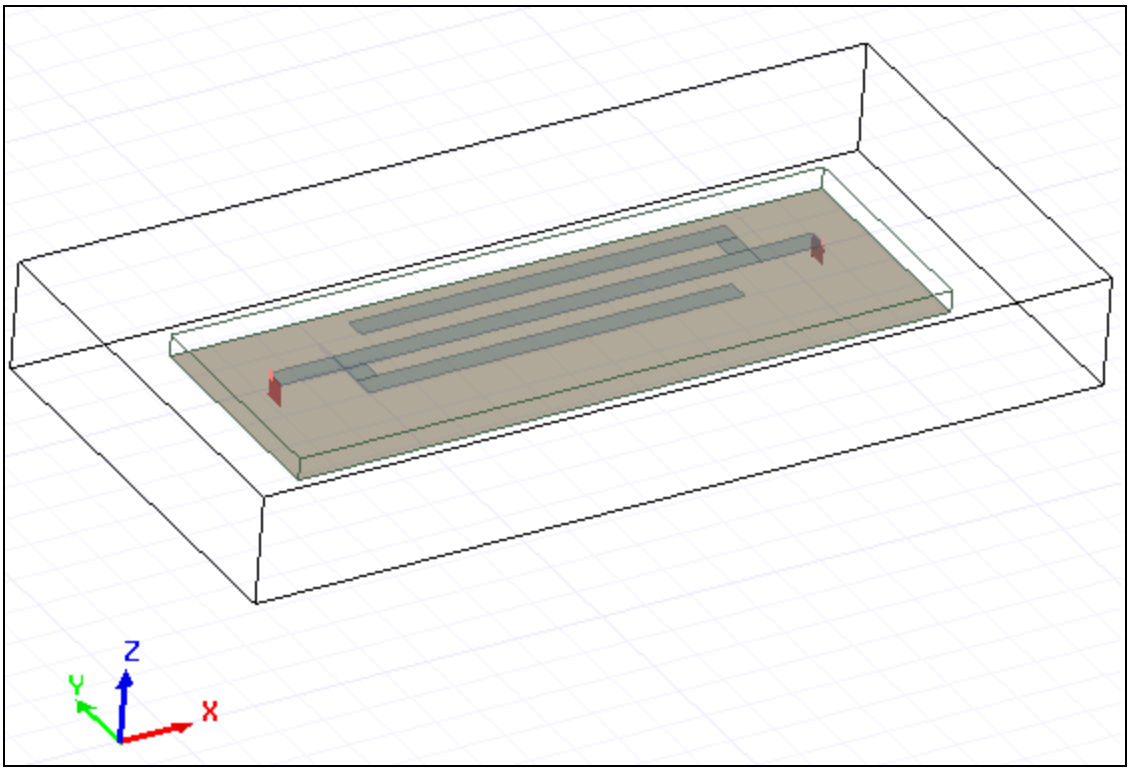


4. Press **Tab** to activate the **Apply** button.

5. Click **Apply** to save changes.
6. Click **OK** to close the **Set HFSS Model Extents** window.
7. Navigate to the **View** ribbon. Then click **HFSS Extents** to display a bounding box around the design in the **Layout Editor**.



8. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.



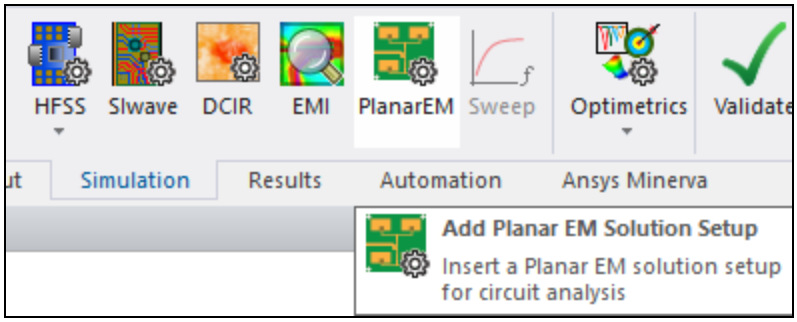
9. Repeat step **6** to hide the bounding box.

Continue to Setting Up a Planar EM Analysis Setup.

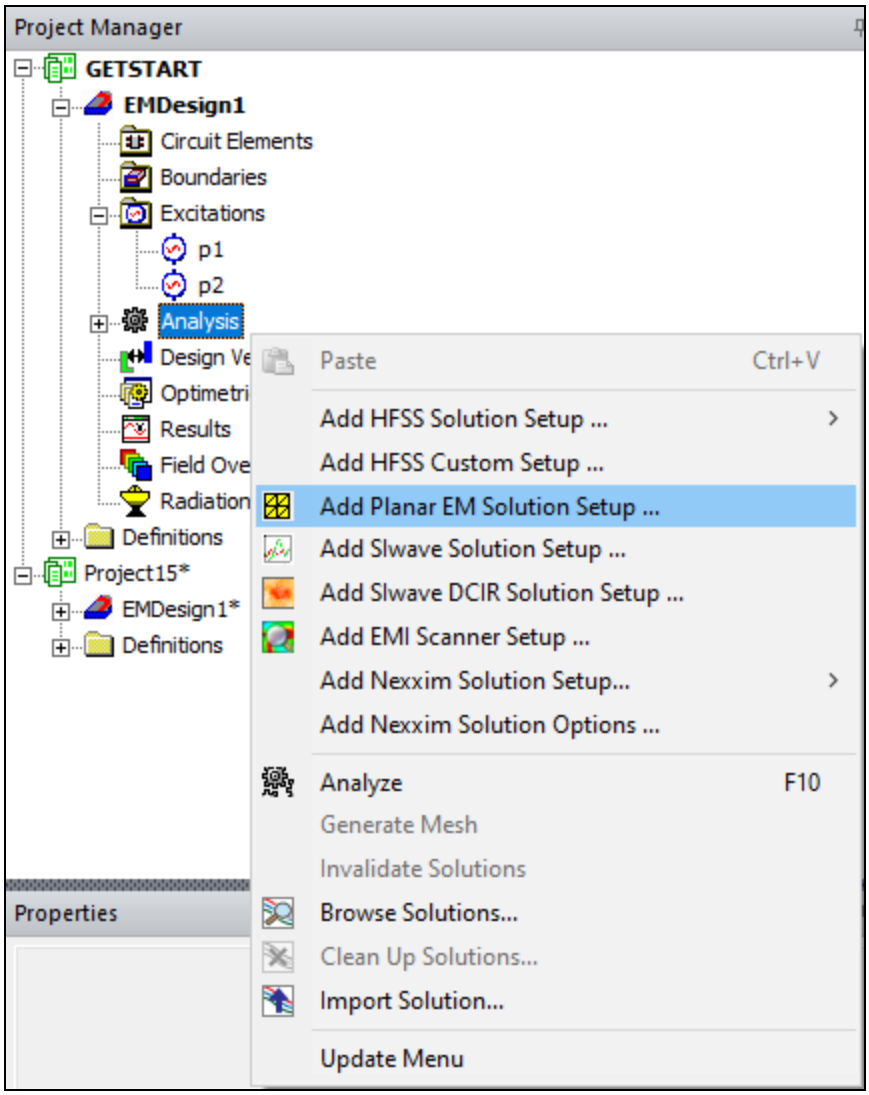
Setting Up a Planar EM Analysis

Solution Setups are listed in the **Project Manager** window (i.e., expand the **Project Tree** > **[*active design folder*]** > **Analysis**). To add a new solution setup to this project, follow these steps.

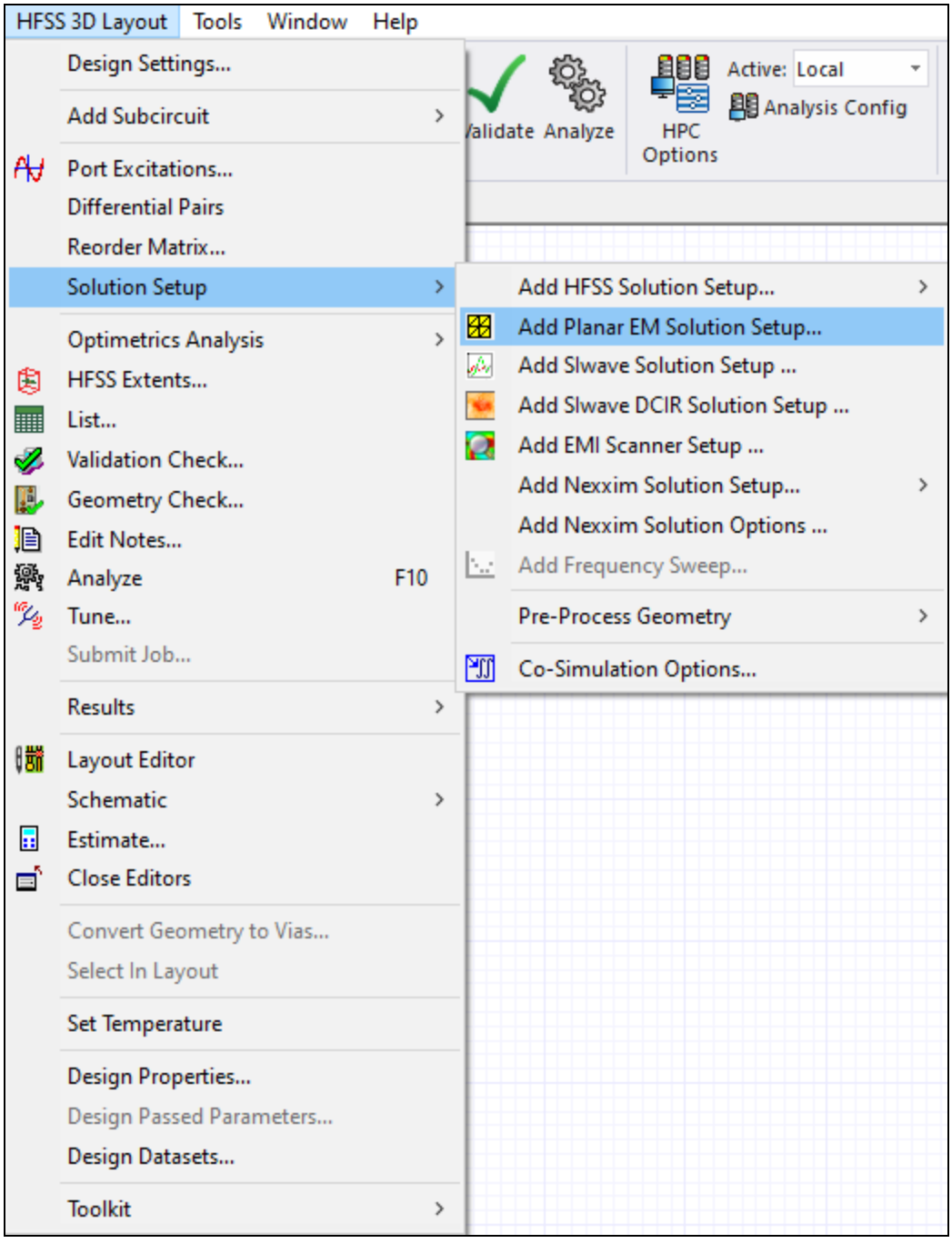
1. Open the **PlanarEMSetup** window by doing one of the following:
   - From the **Simulation** ribbon tab, click **PlanarEM** (**Add Planar EM Solution Setup**).



   - Right-click **Analysis** in the **Project Manager** window and click **Add Planar EM Solution Setup** .

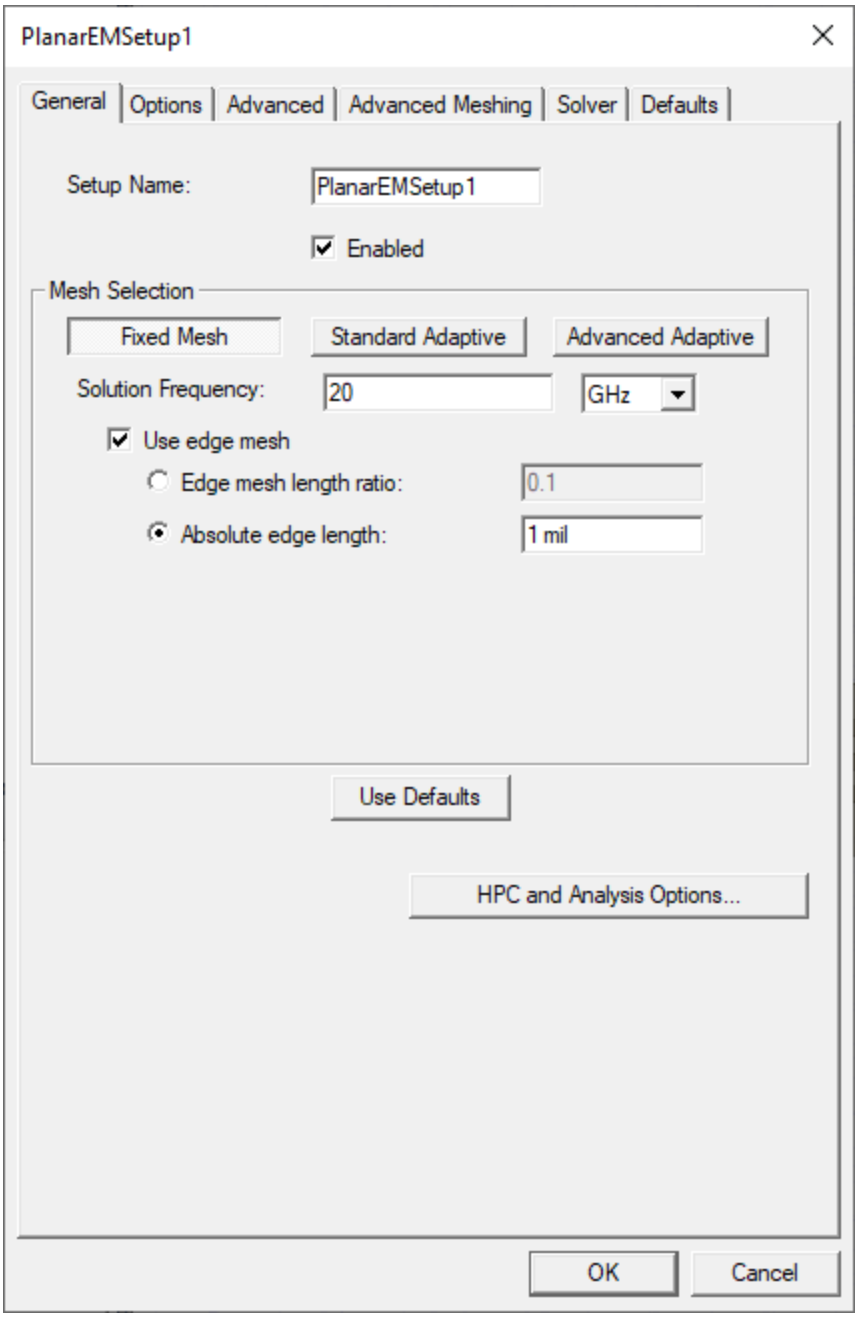


- From **HFSS 3D Layout**, select **Solution Setup** > **Add Planar EM Solution Setup**.

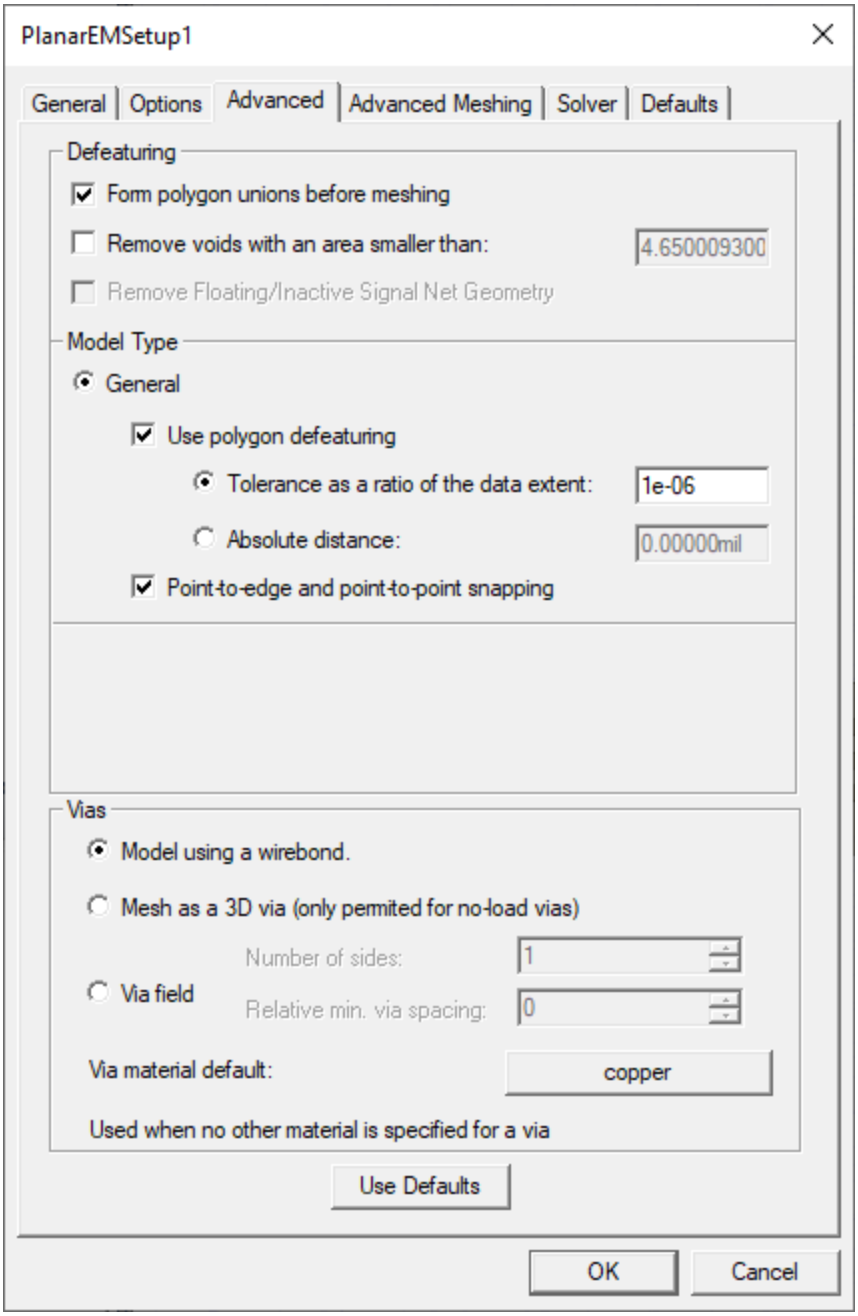


2. From the **PlanarEMSetup** window > **Mesh Selection** area, do the following:
   - Ensure **Fixed Mesh** is selected.
   - Enter **20** in the **Solution Frequency** field.

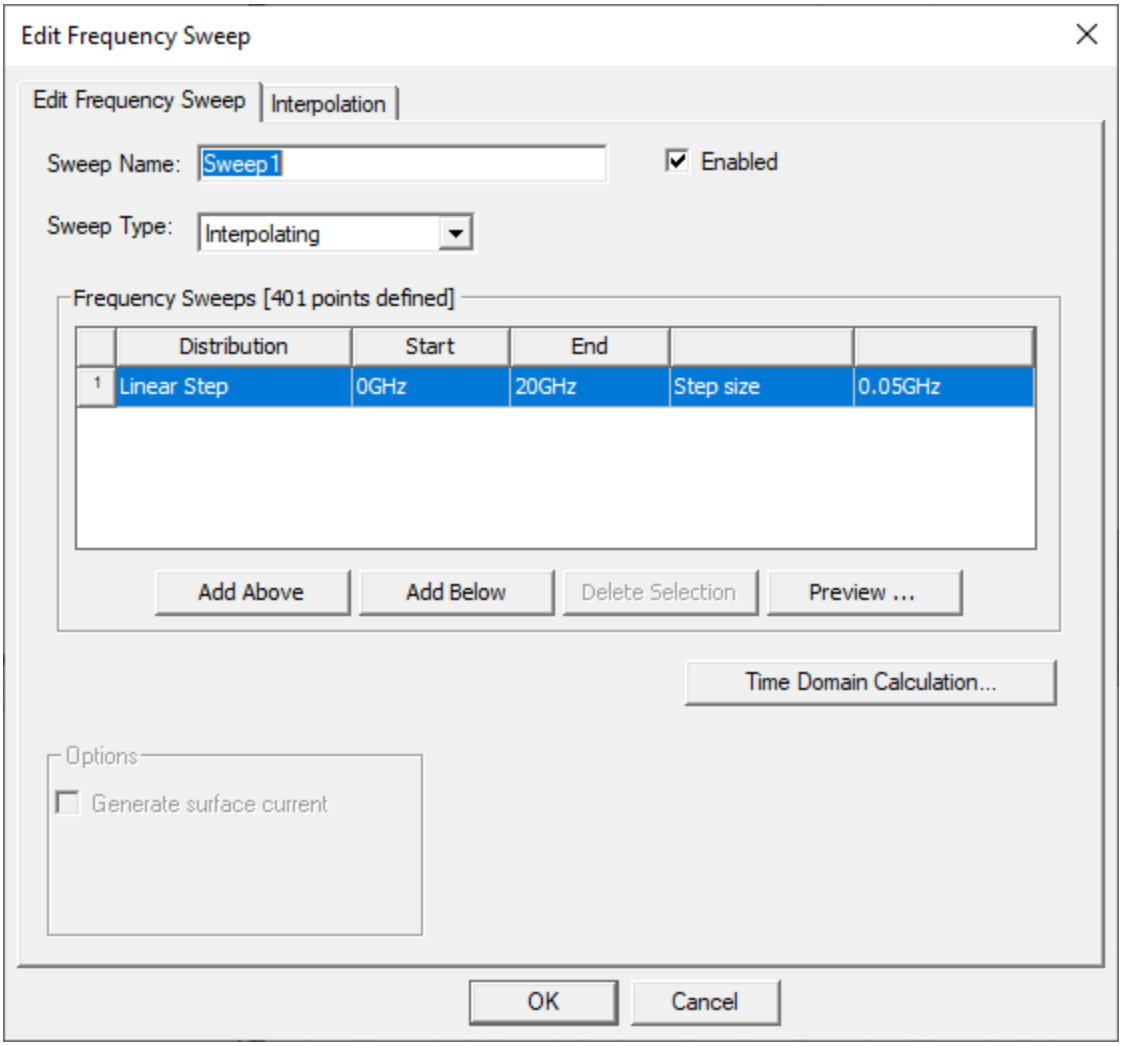
- Check the **Use edge mesh** check box.
- Select **Absolute edge length** and enter **1 mil** in the field.



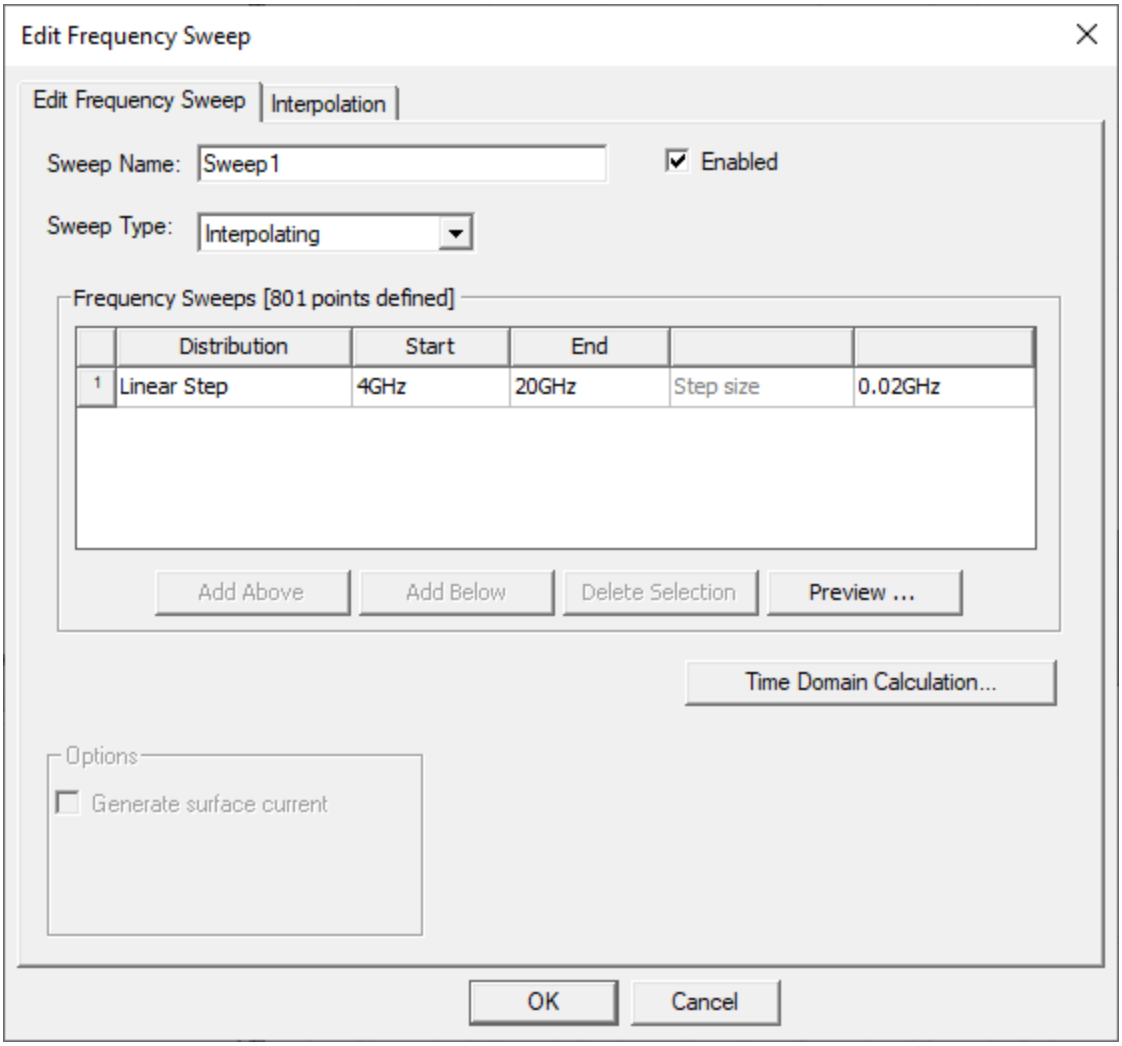
3. Navigate to the **Advanced** tab.



4. From the **Defeaturing** area, ensure the **Form polygon unions before meshing** box is checked.
5. Click **OK** to close the **PlanarEMSetup** window and open the **Edit Frequency Sweep** window.



6. Ensure **Interpolating** is selected from the **Sweep Type** drop-down menu.
7. Ensure **Linear Step** is selected from the **Distribution** drop-down menu.
8. Enter the following parameters in the first row of the **Frequency Sweeps** table:
    - **4.0** (GHz) in the **Start** field.
    - **20.0** (GHz) in the **End** field.
    - **0.02** (GHz) in the **Step size** field.



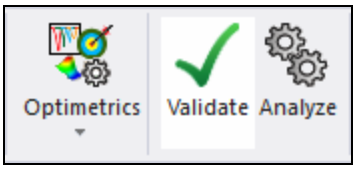
9. Click **OK** to add the interpolating sweep and close the **Edit Frequency Sweep** window.

Continue to [Validating and Analyzing the Design](#).

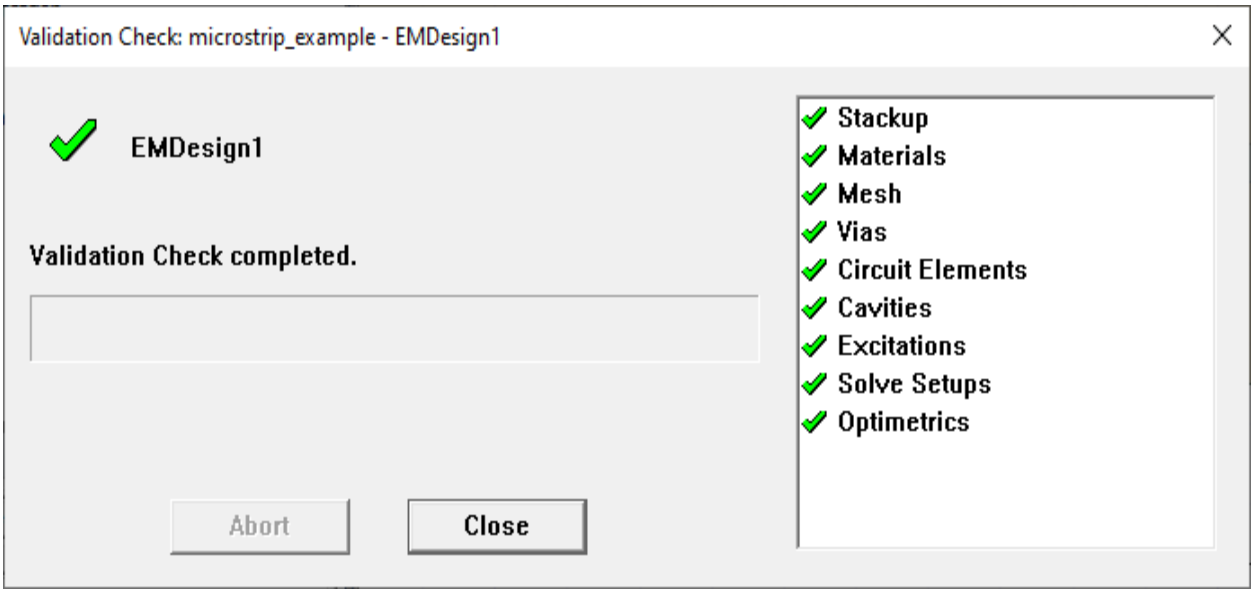
Validating and Analyzing the Design

Complete these steps to validate the active design and analyze both the HFSS and Planar EM setups.

1. From the **Simulation** ribbon tab, click **Validate** to open the **Validation Check** window.

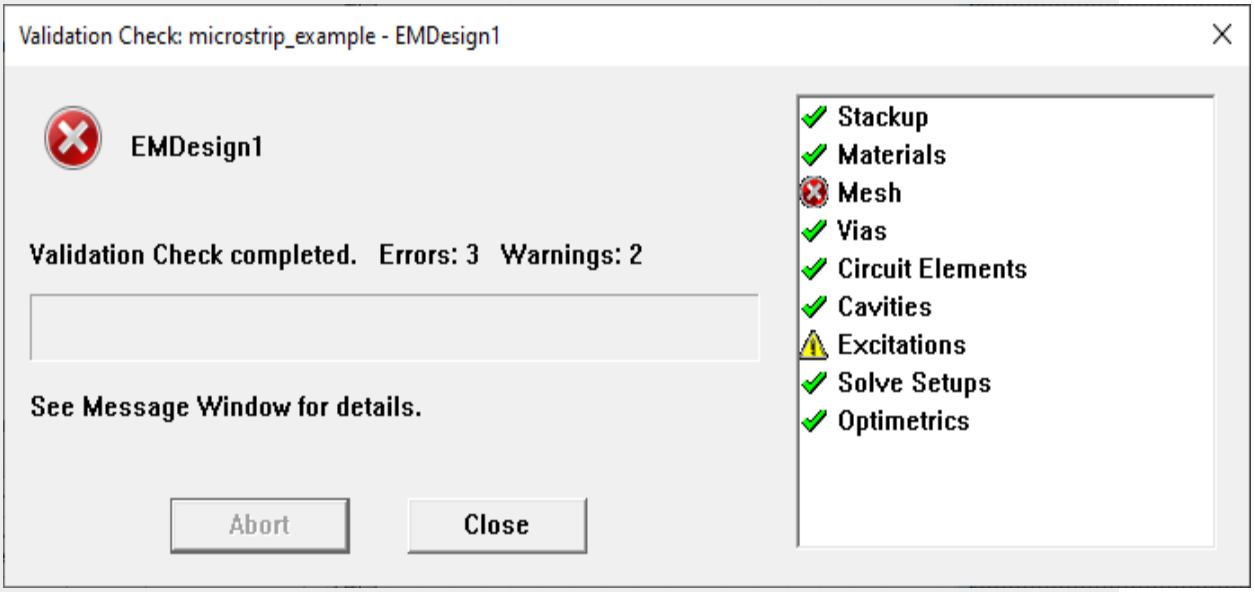


Assuming the design is valid, the **Validation Check** window will display the following result.



2. Click **Close**.

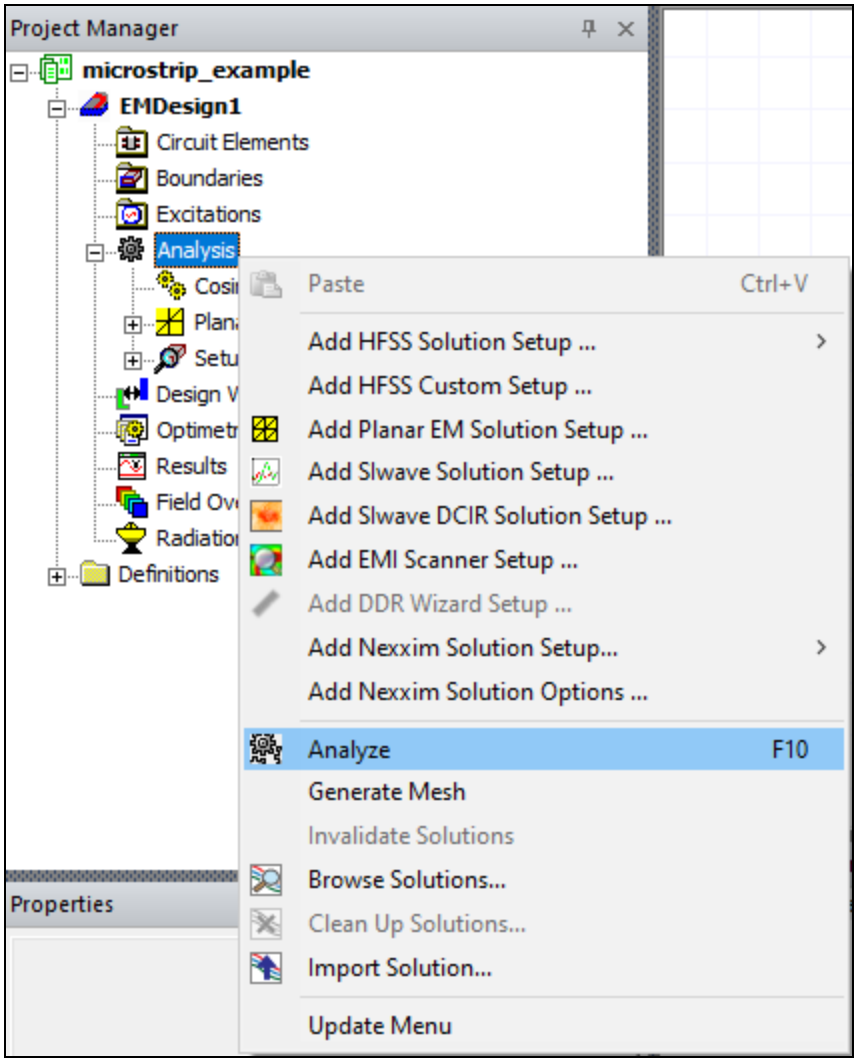
> **Note:**
>
> If the validation check detects any errors or warnings, view detailed feedback in the **Message Manager** window.



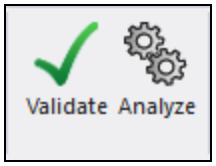
3. Solve both the HFSS Analysis and Planar EM Analysis setups, and their associated sweeps, by doing one of the following:

> **Note:**
>
> To run an individual analysis, either select the setup in the **Project Manager** window and click **Analyze** in the **Simulation** ribbon, or right-click the setup and select **Analyze**.

- Right-click **Analysis** in the **Project Manager** window and select **Analyze**.



- From the **Simulation** ribbon tab, click **Analyze**.

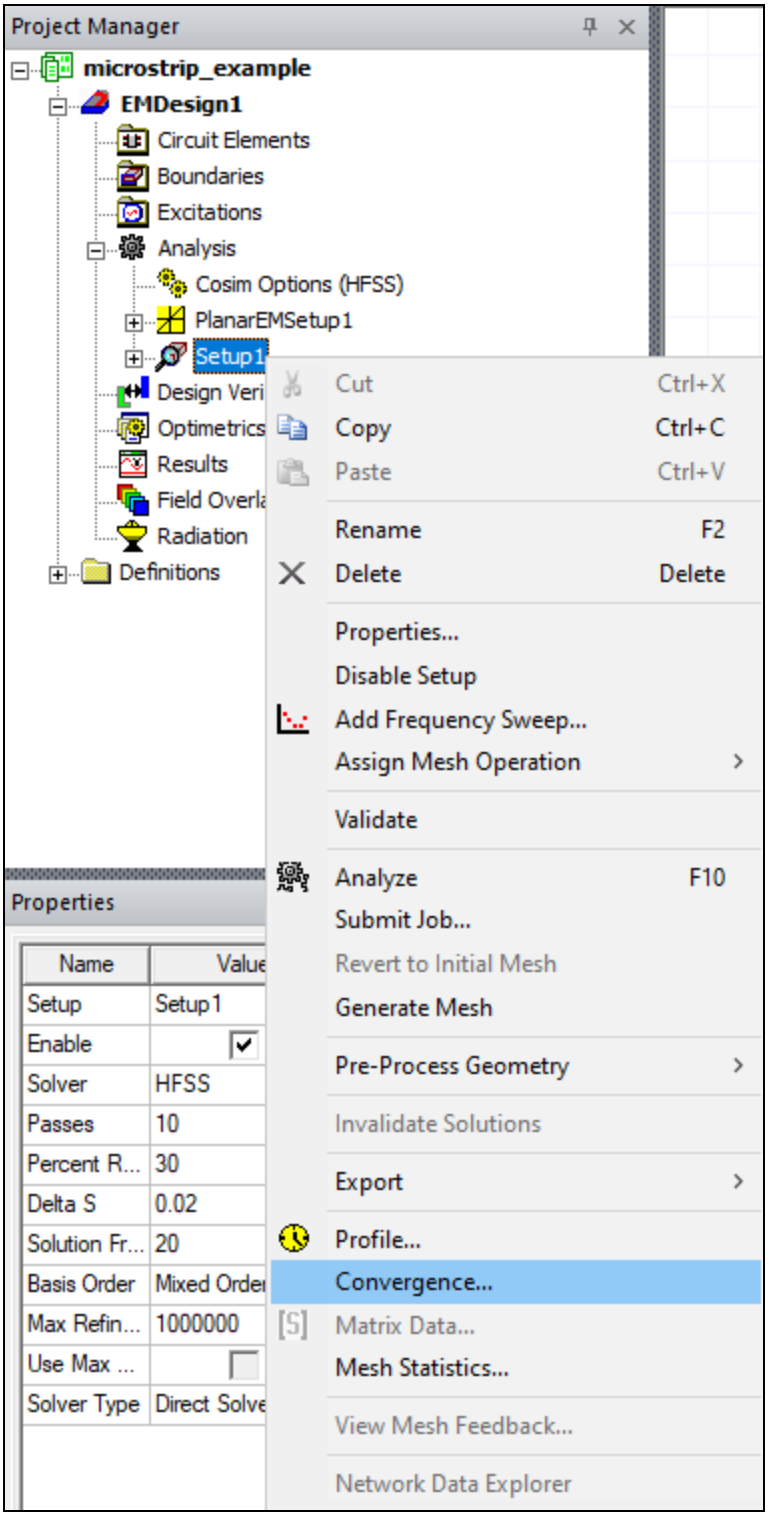


Continue to Viewing Convergence History.

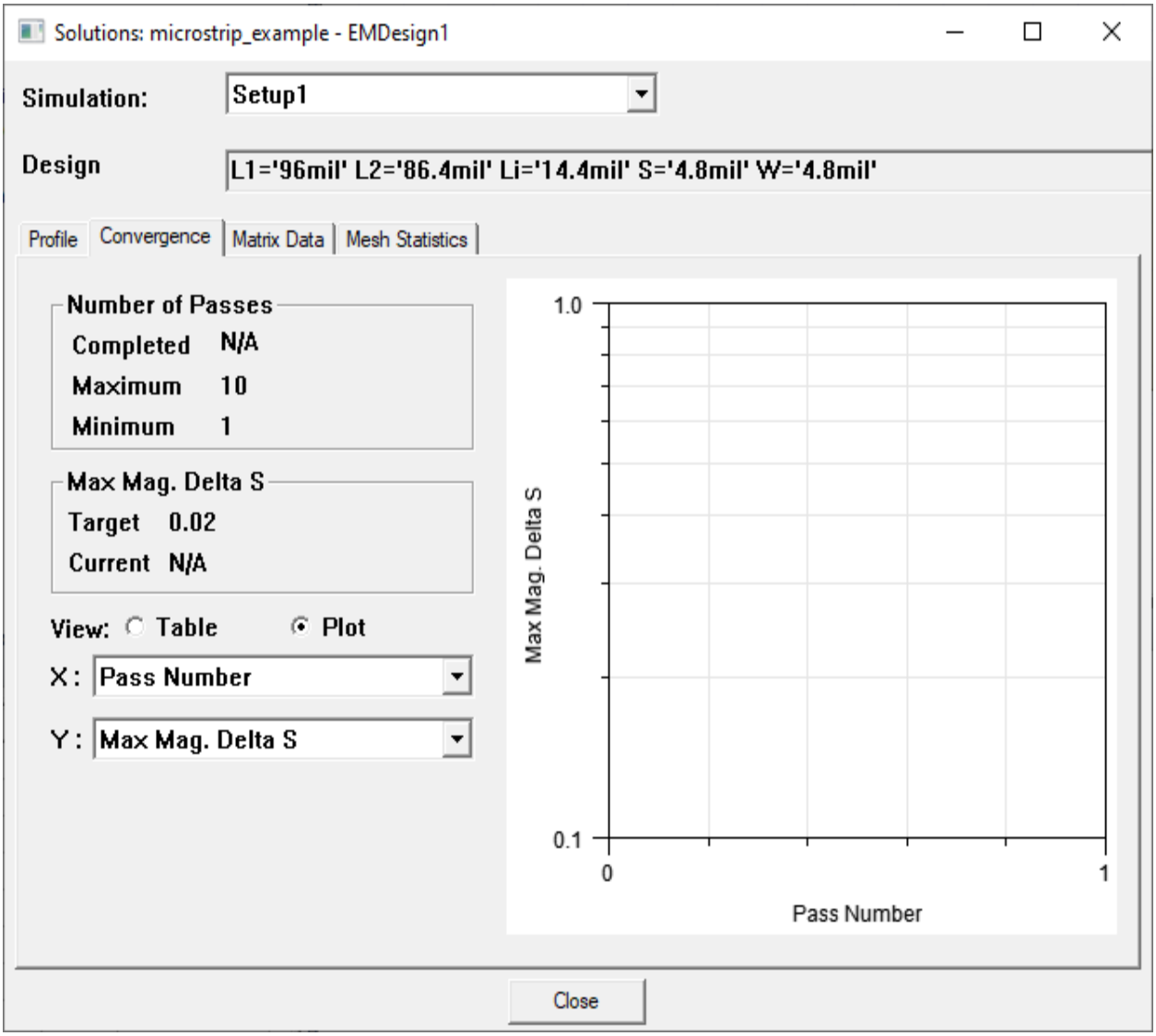
Viewing Convergence History

Complete these steps to view the active design's analysis convergence history.

1. From the **Project Manager** window, right-click the HFSS Setup (i.e., **Setup1**) and select **Convergence** to open the **Solutions** window.



2. Select **Plot**.



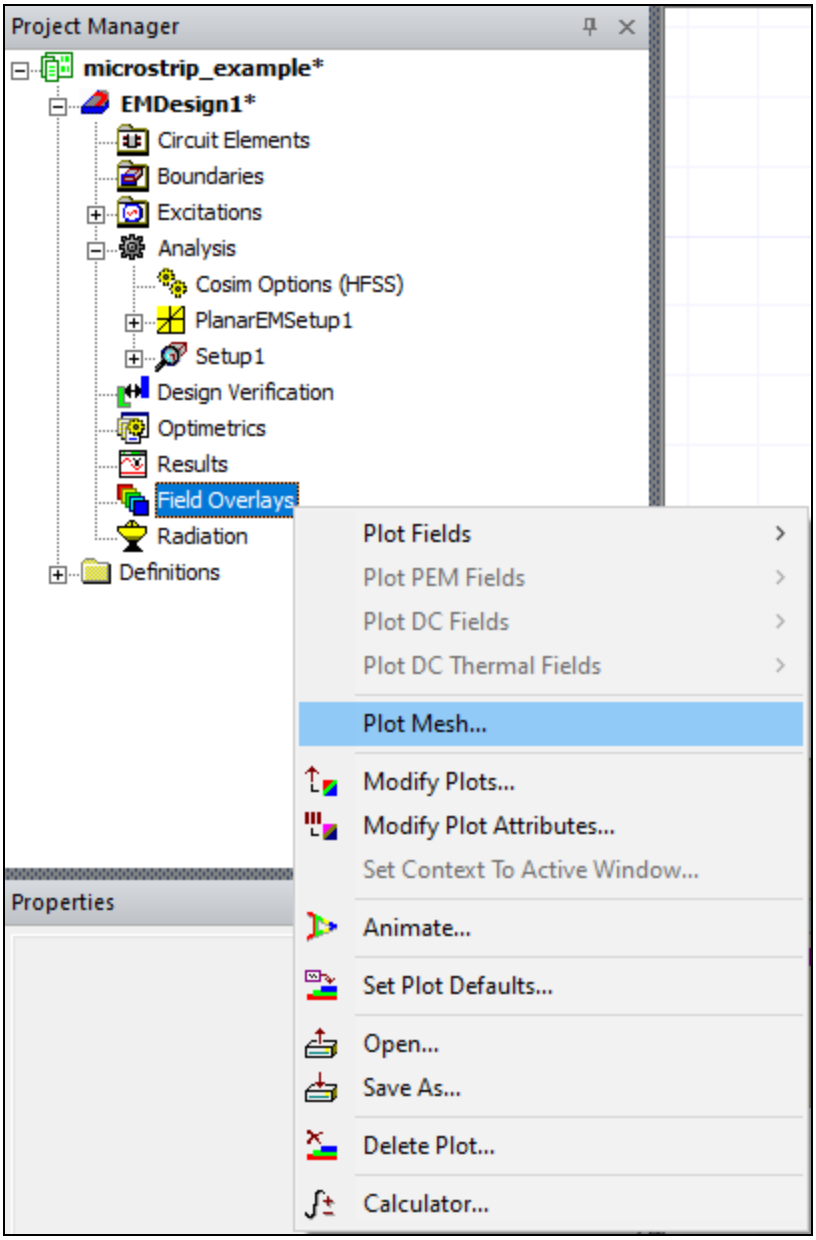
3. Click **Close**.

Continue to Plotting the HFSS Mesh.

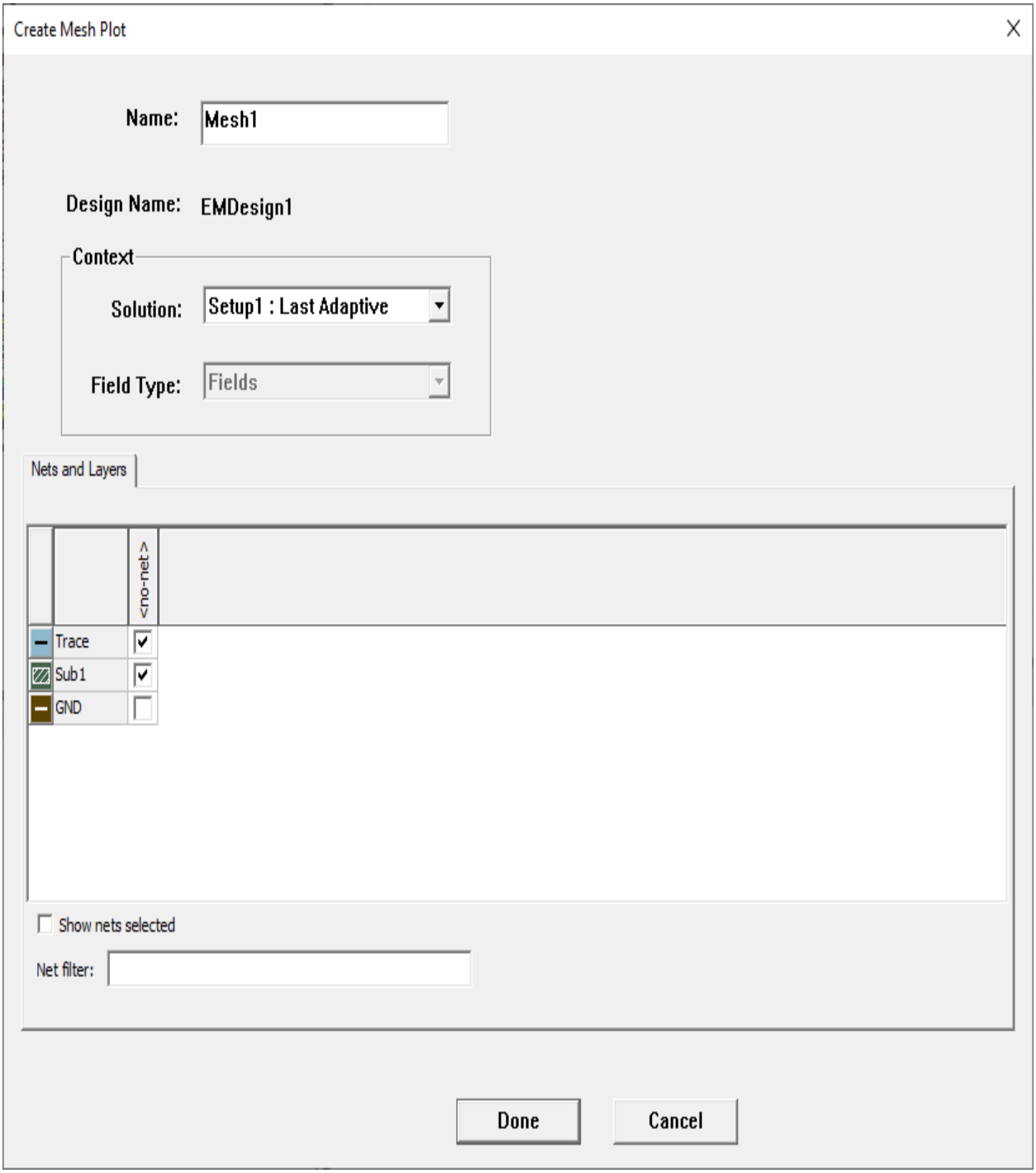
Plotting the HFSS Mesh

Complete these steps to plot and view a mesh of the HFSS Setup solution.

1. From the **Project Manager** window, right-click **Field Overlays** and select **Plot Mesh** to open the **Create Mesh Plot** window.



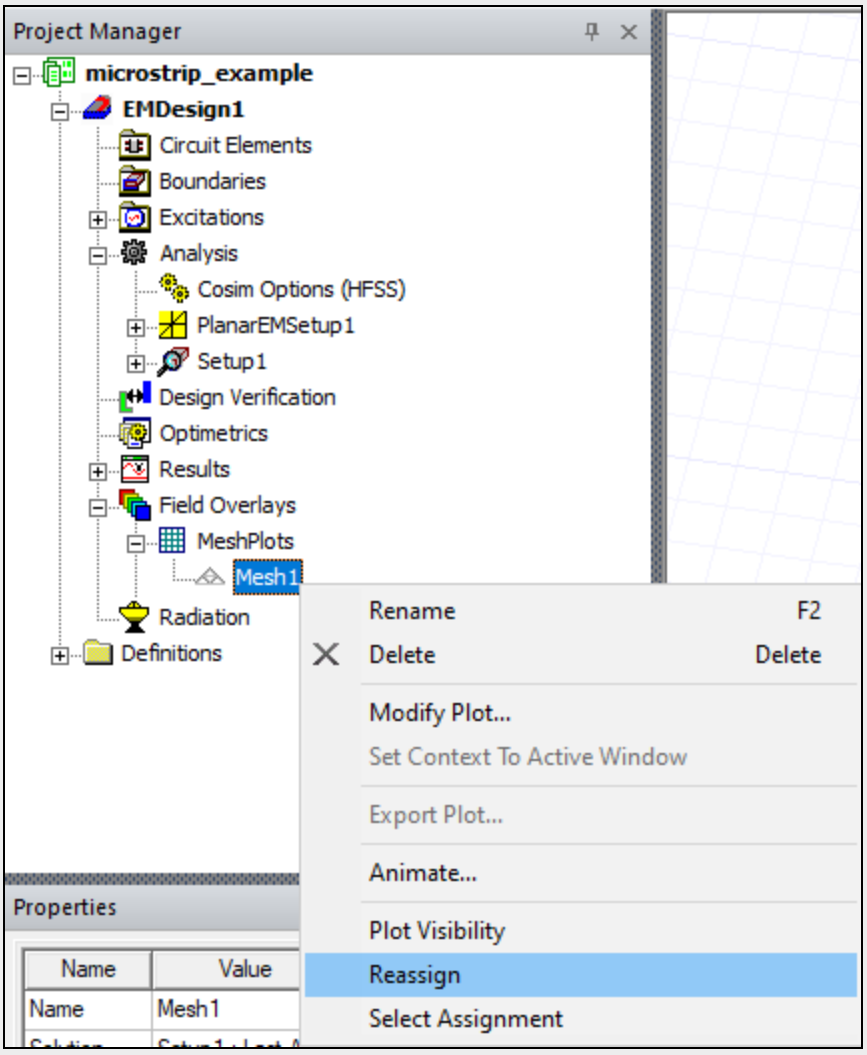
2. Under the **Nets and Layers** tab, check the **Trace** and **Sub1** boxes to only plot the mesh on those layers.



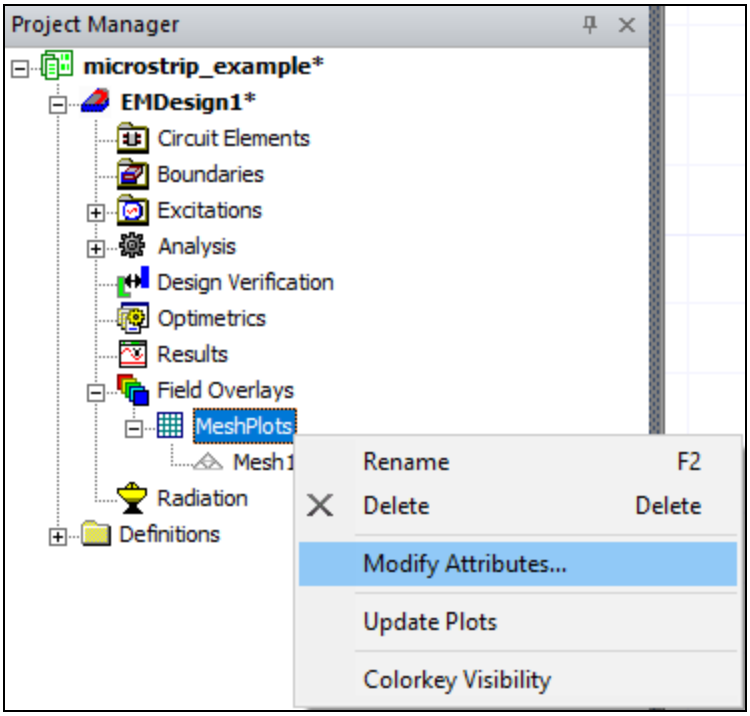
3. Click **Done** to close the **Create Mesh Plot** window and display the mesh for the **Setup1 : Last Adaptive** solution in the **Layout Editor**.

Note:

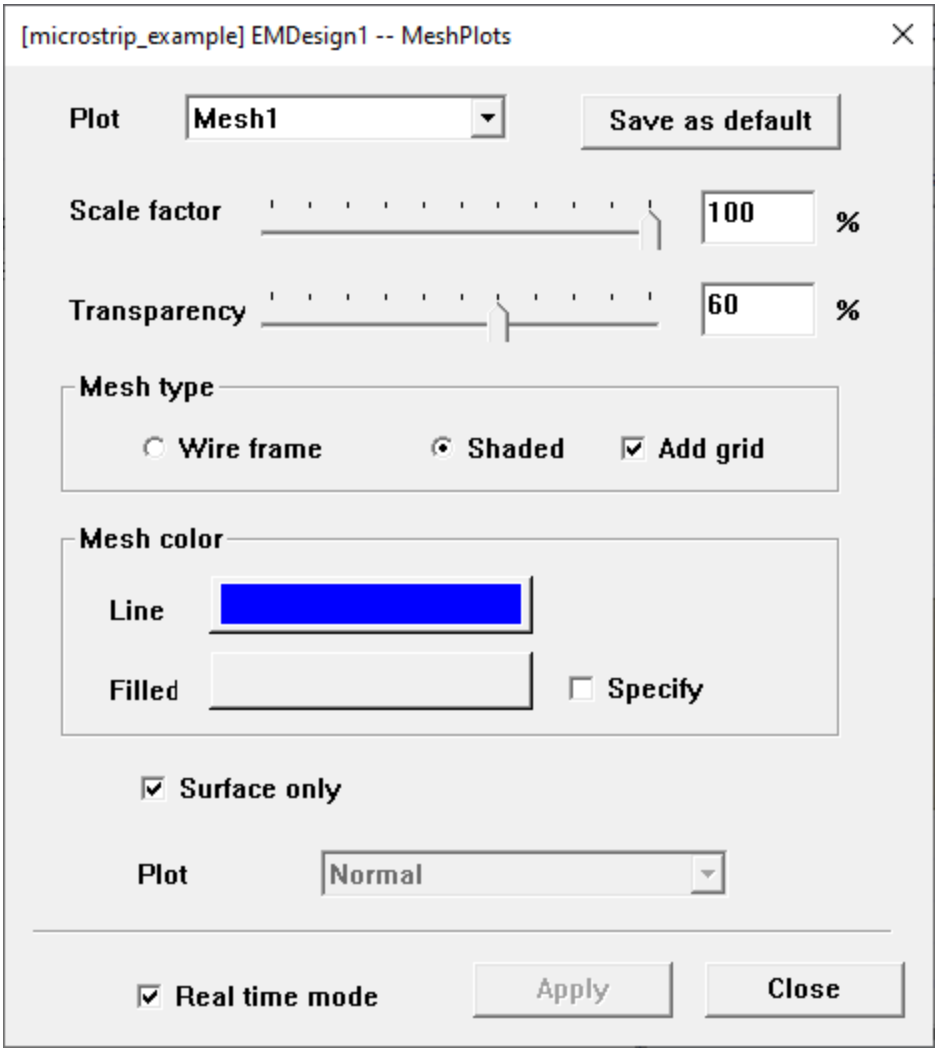
To change a mesh, from the **Project Manager** window, expand **Field Overlays** > **MeshPlots**. Then right-click the mesh (e.g., **Mesh1**) and select **Reassign**.



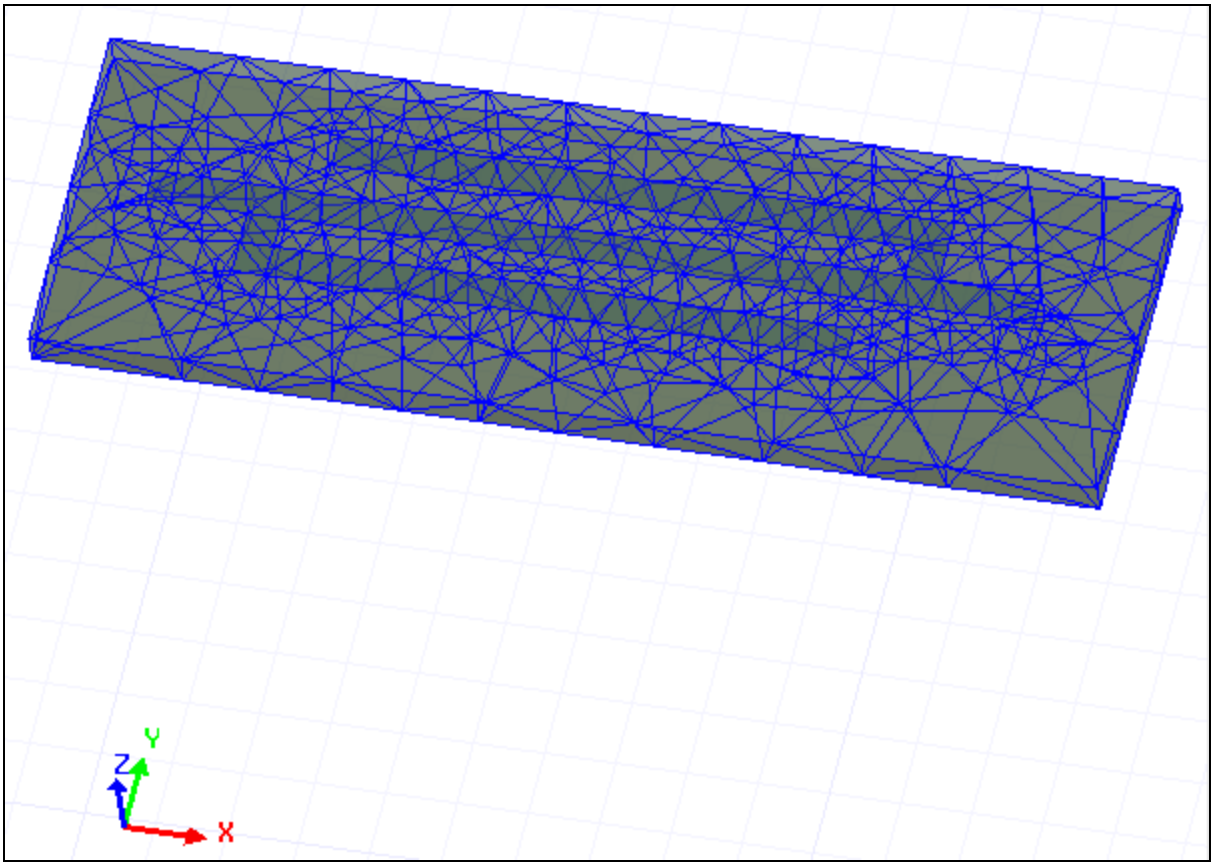
4. From the **Project Manager** window, expand **Field Overlays**. Then right-click **MeshPlots** and select **Modify Attributes** to open the **MeshPlots** window.



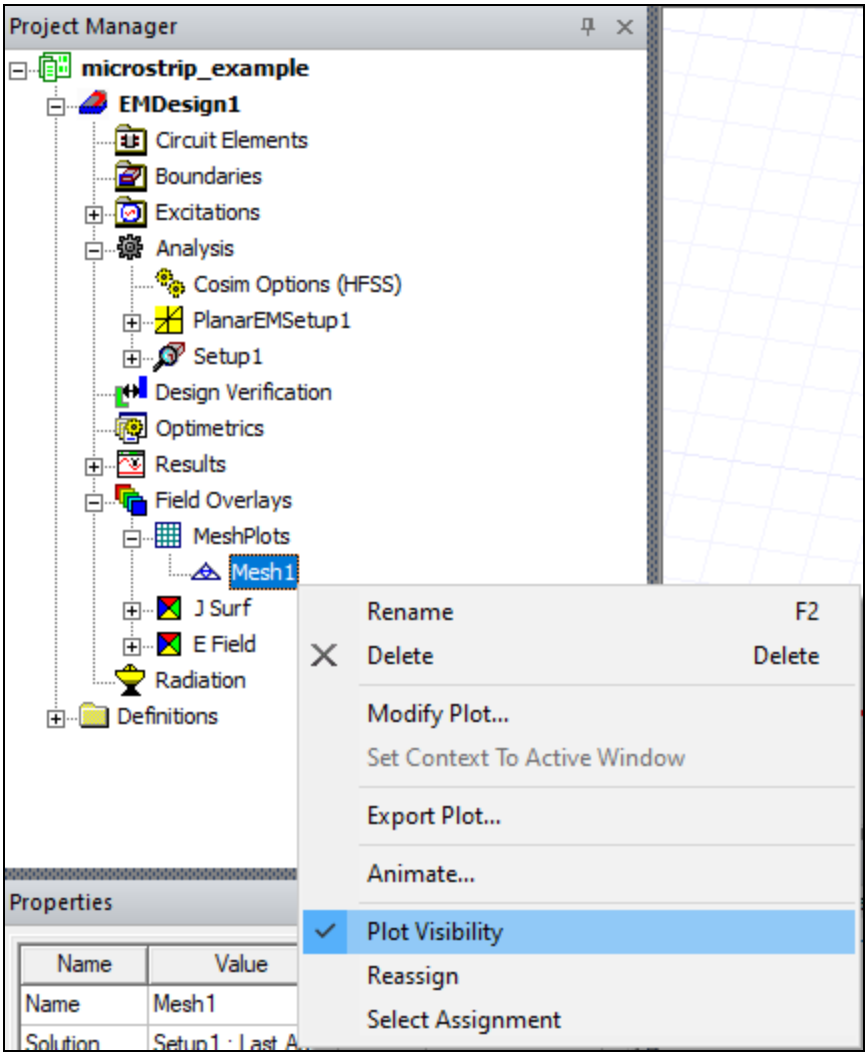
5. Either **click+drag** the **Transparency** scale marker or enter **60** in the field to improve the visibility of the mesh lines (i.e., element edges).



6. Click **Apply**. Then click **Close** to return to the **Layout Editor**.
7. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.



8. From the **Project Manager** window, right-click the mesh (e.g., **Mesh1**) and select **Plot Visibility** to remove the adjacent check mark and hide the mesh.

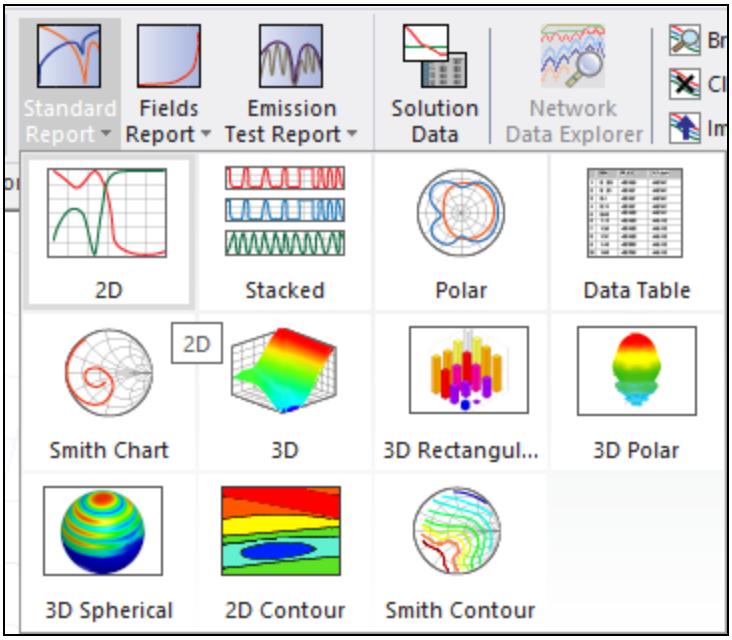


Continue to [Creating the S-Parameter Plot].

Creating the S-Parameter Plot

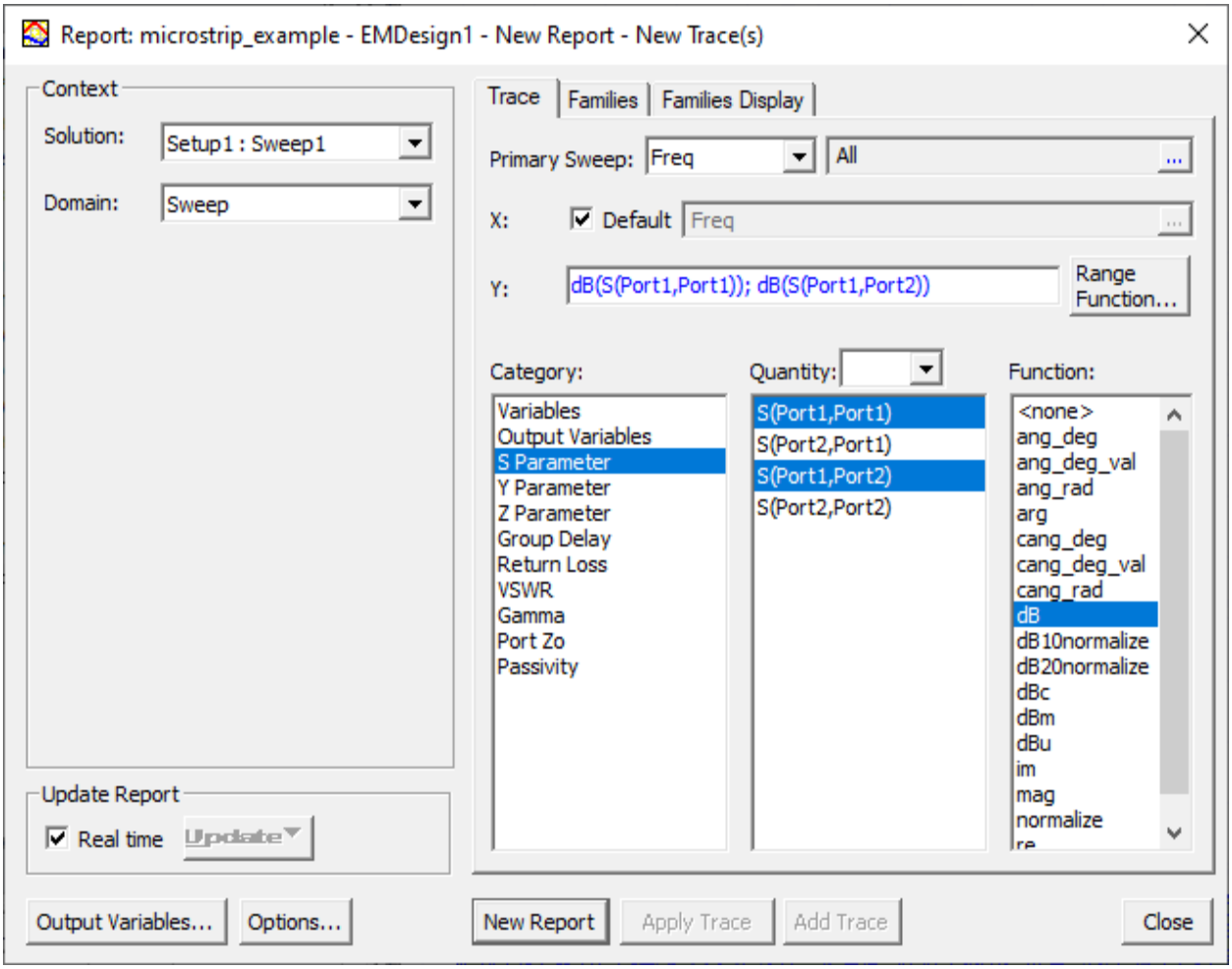
Complete these steps to create an S-Parameter plot with four traces, two each, from the HFSS and Planar EM analyse, then compare the result of the solution types.

1. From the **Results** ribbon tab, select **Standard Report** > **2D** to open the **Report** window.

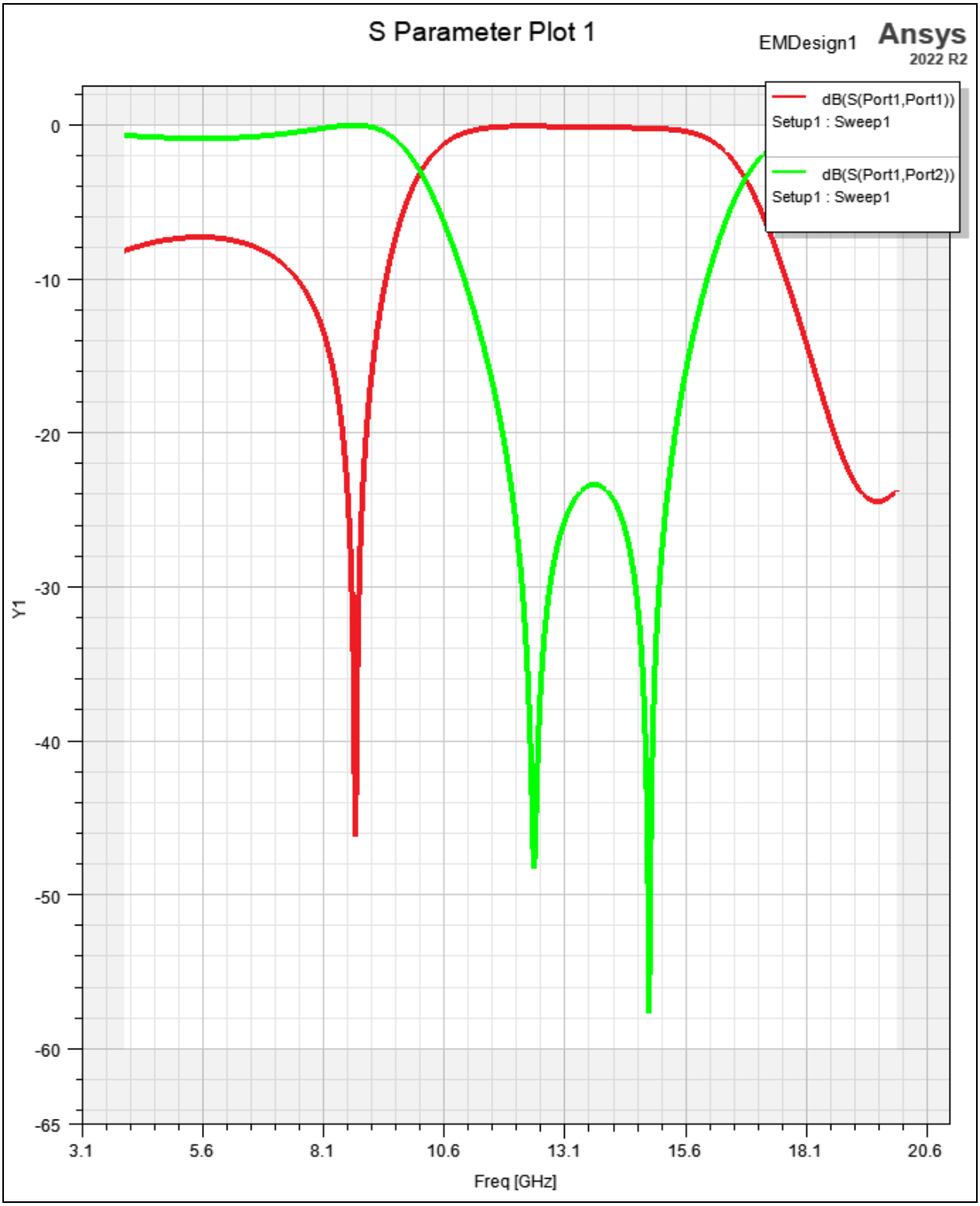


2. Ensure the following settings are selected:
   - From the **Context** area, **Setup1 : Sweep1** is selected from the **Solution** drop-down menu.
   - **Sweep** is selected from the **Domain** drop-down menu.
   - From the **Trace** tab, **S Parameter** is selected from the **Category** list.
   - **dB** is selected from the **Function** list.

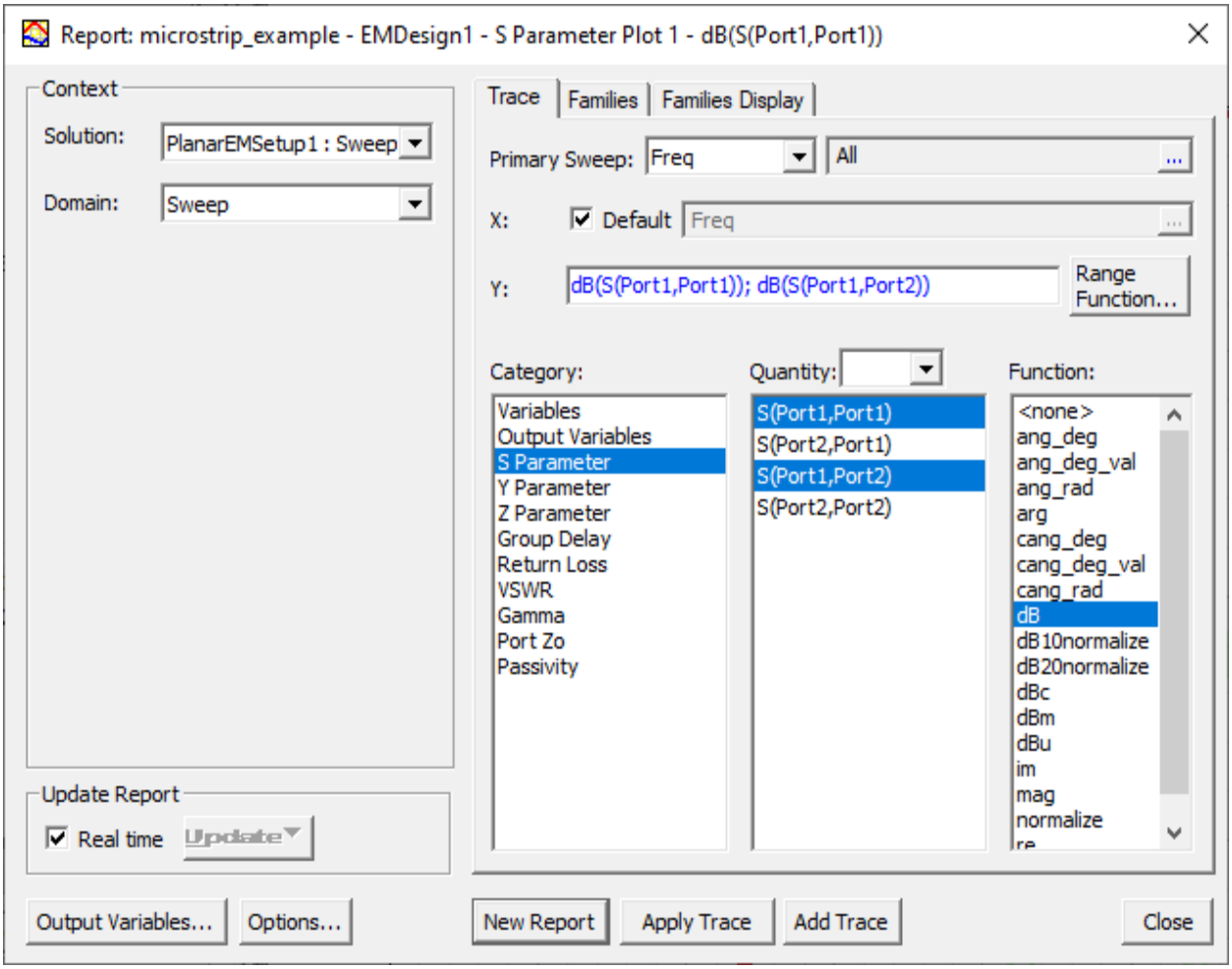
3. **Ctrl+click** to select **S(Port1,Port1)** and **S(Port1,Port2)** from the **Quantity** list.



4. Click **New Report** and the following plot appears. However, do **not** close the **Report** window.

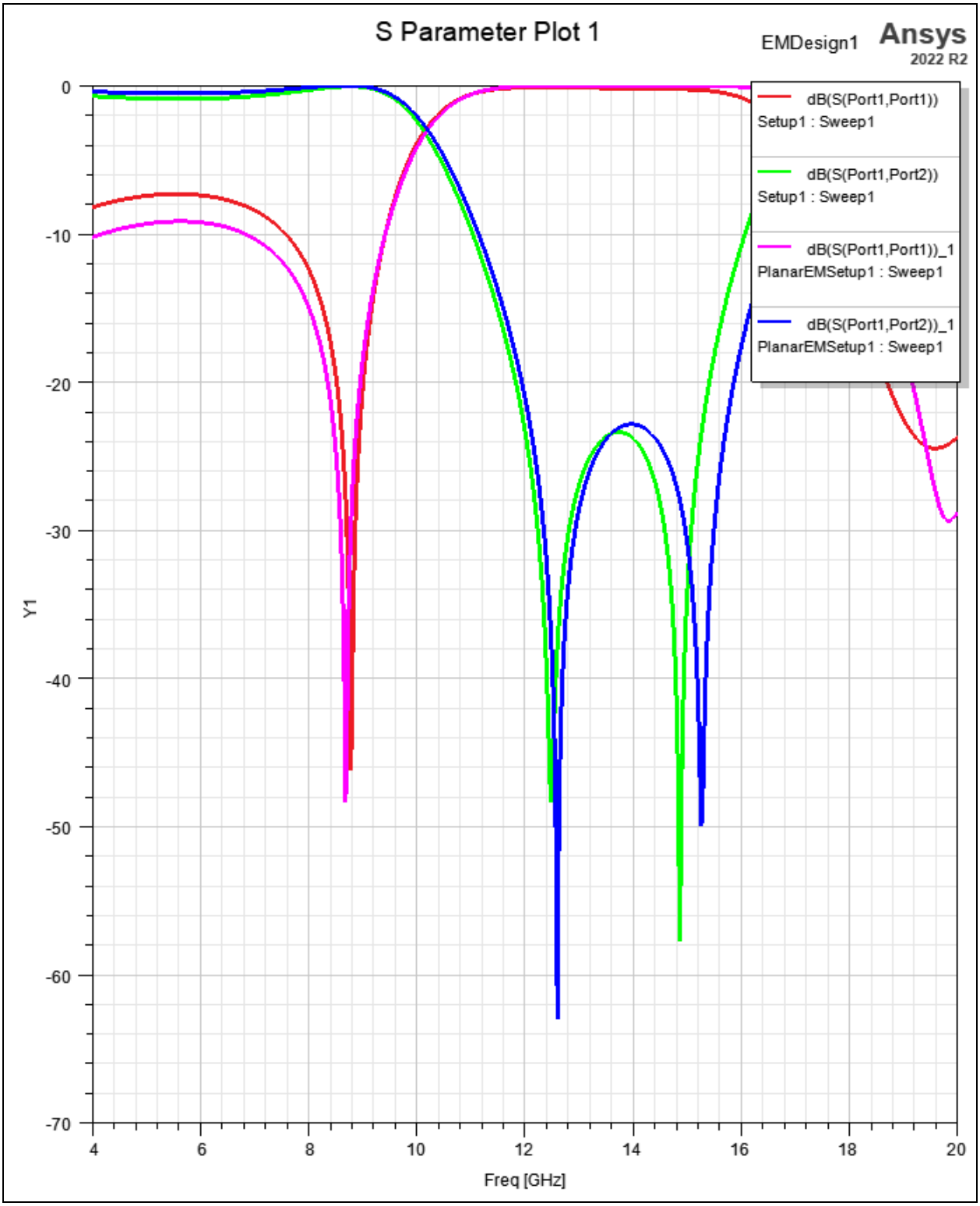


5. From the **Context** area, select **PlanarEMSetup1 : Sweep1**from the **Solution** drop-down menu.



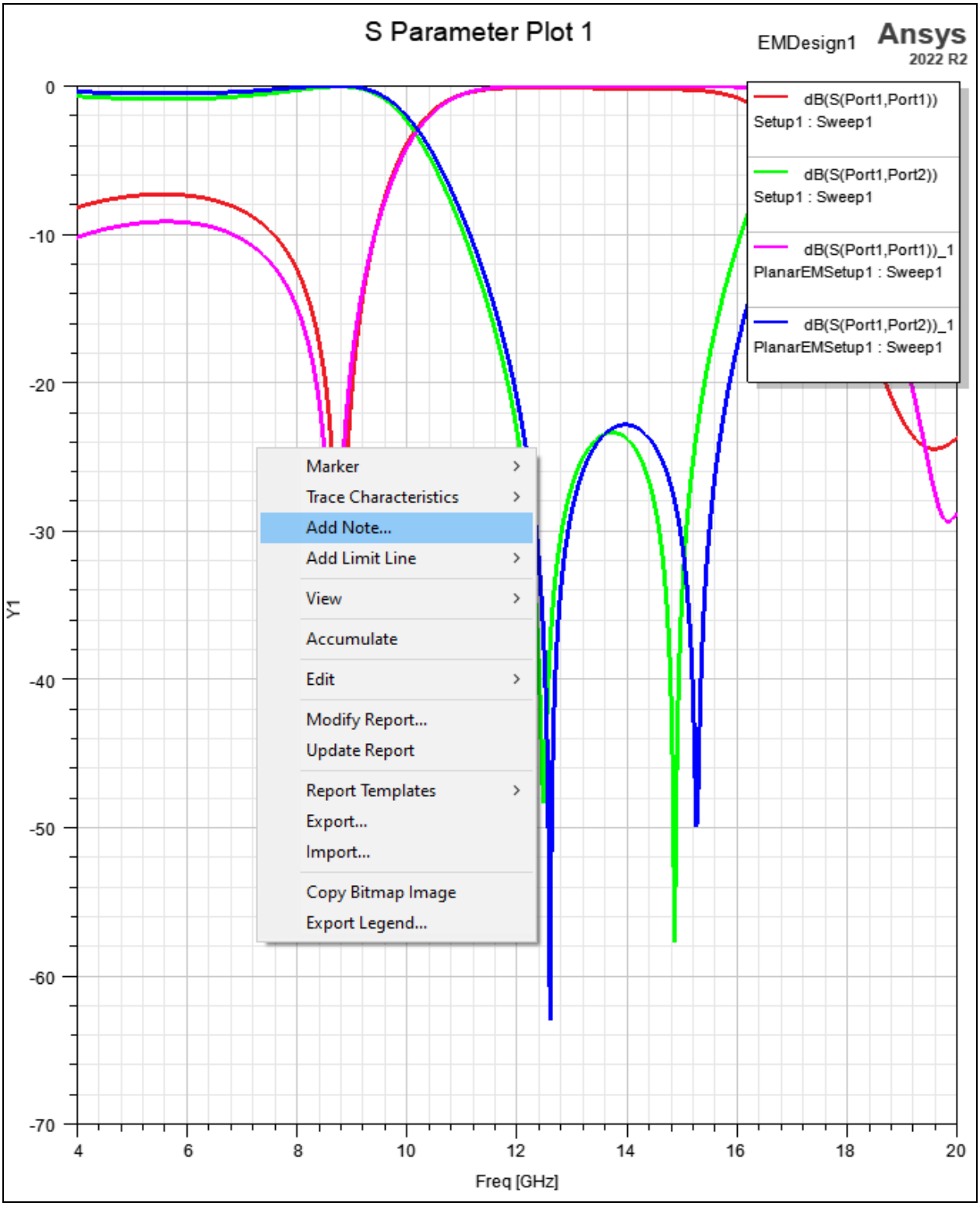
6. Click **Add Trace**.

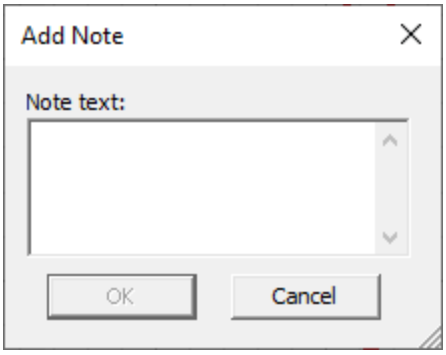
7. Click **Close** to view the new plot.



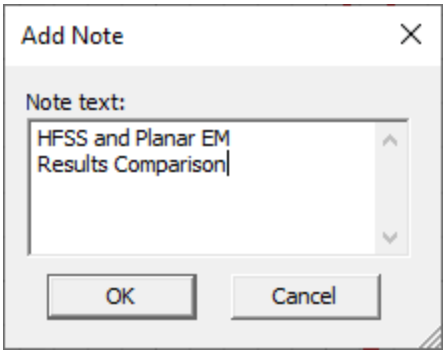
8. Click elsewhere in the **View** window to deselect the traces.

9. Right-click within the **View** window and select **Add Note** to open the **Add Note** window.

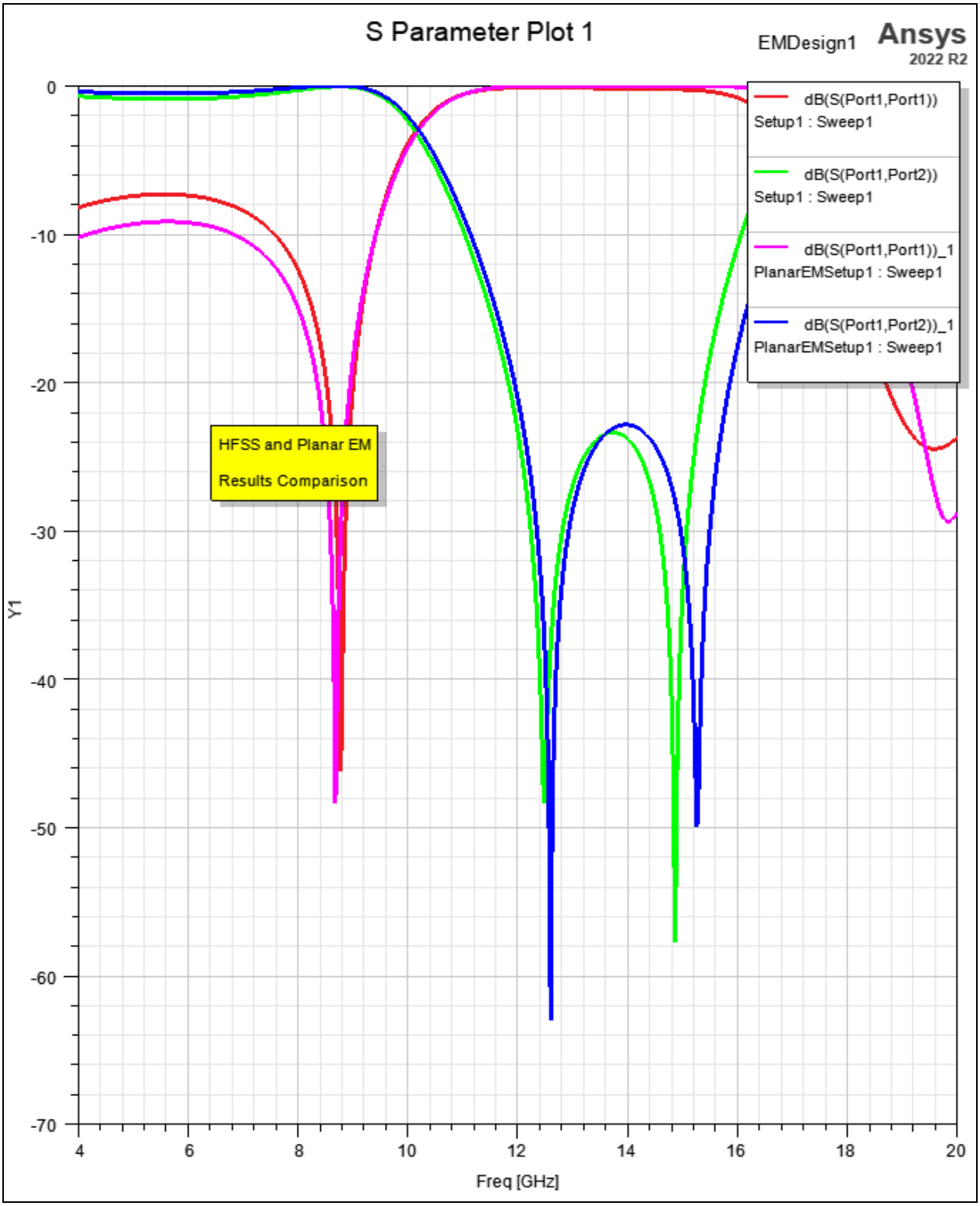




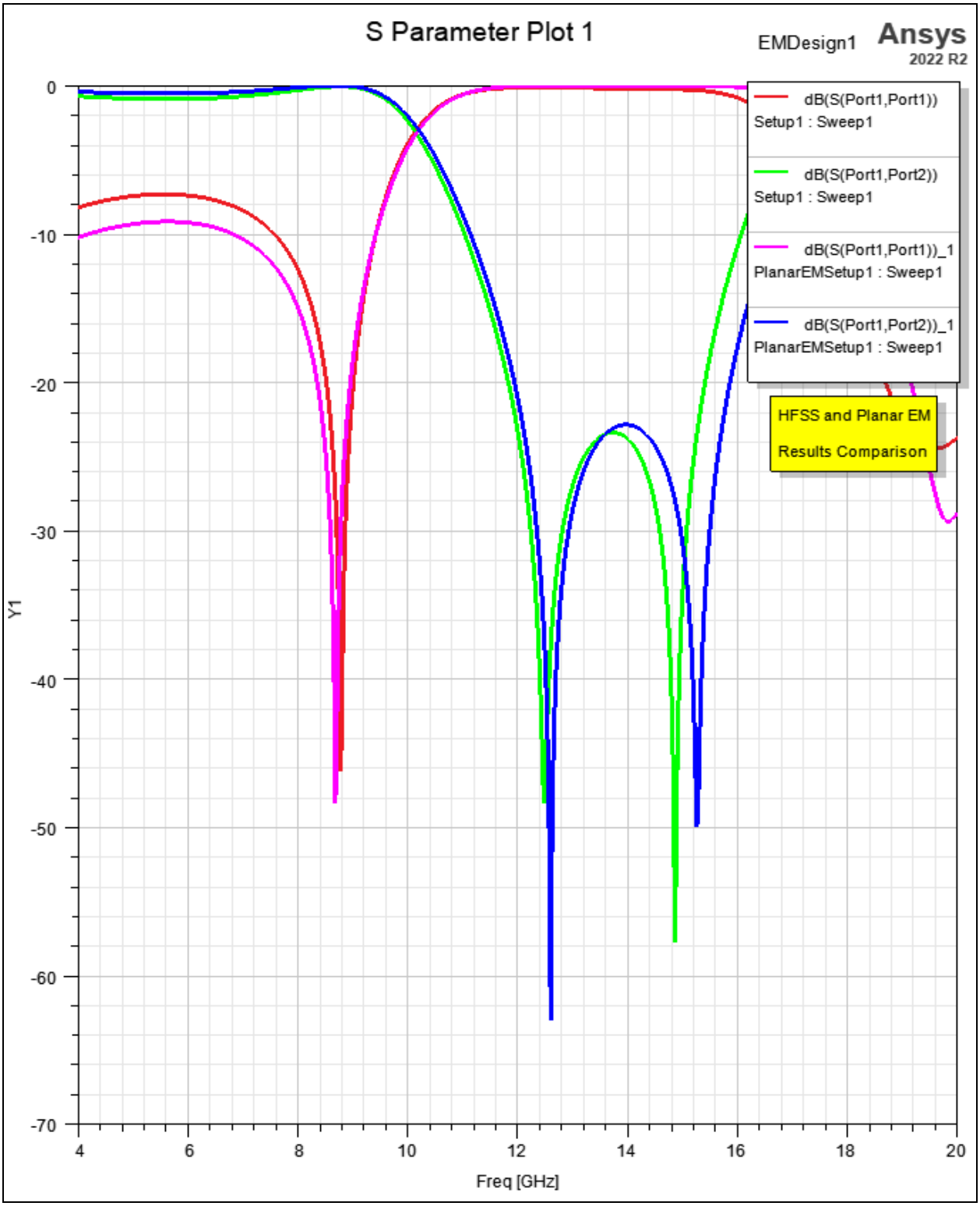
10. Enter the following note in the **Add Note** field:
    - Type **HFSS and Planar EM** and press **Enter**.
    - Type **Results Comparison**.



11. Click **OK** to close the **Add Note** window and add the note to the plot.



12. Drag the note and legend to a desirable location.



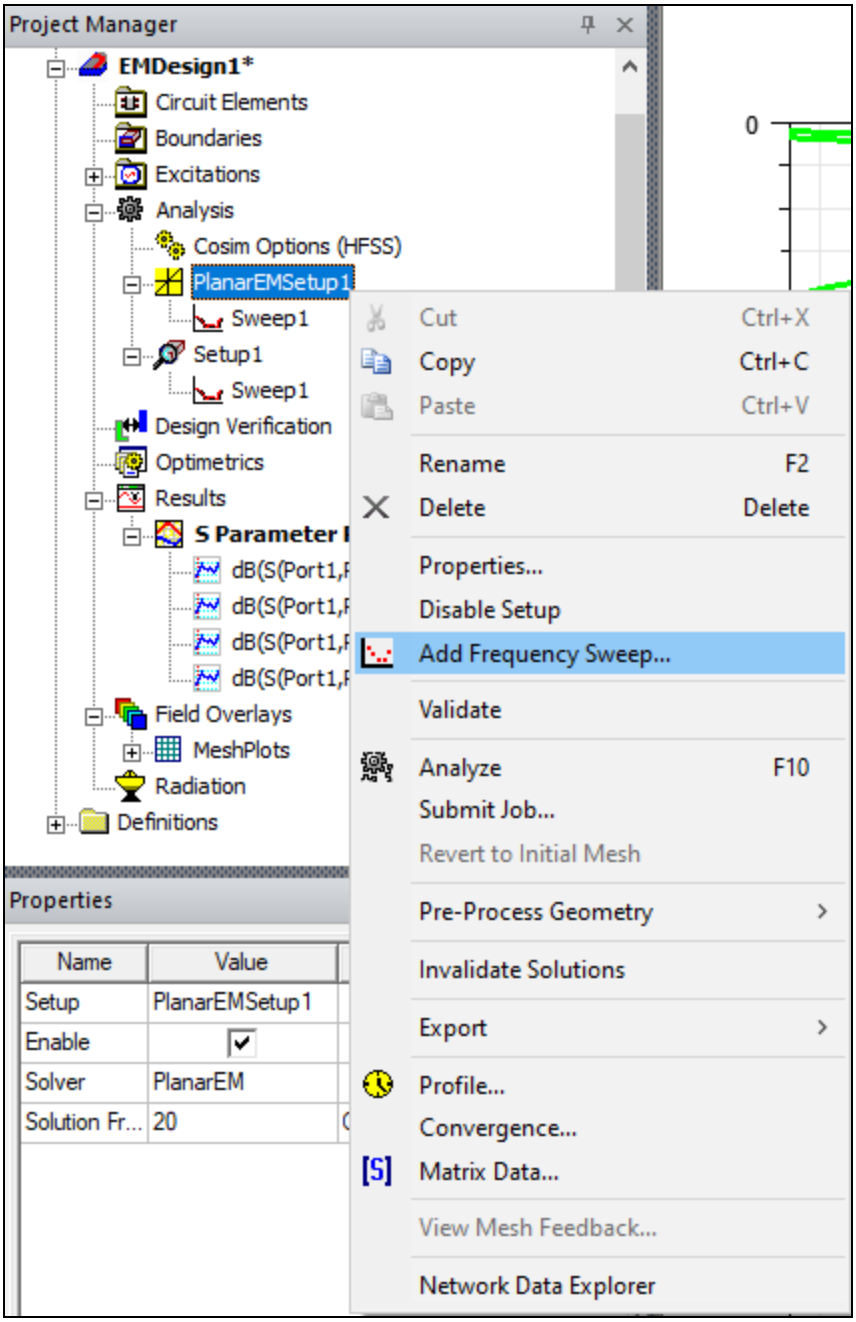
> **Note:**
>
> Both solutions demonstrate the bandstop behavior over the range of about 10 to 17 GHz. The only significant difference is in the magnitude of the three minimal points along the traces (at 8.75, 12.5, and 15.2 GHz).

Continue to Adding and Analyzing a Discrete Sweep.

Adding and Analyzing a Discrete Sweep

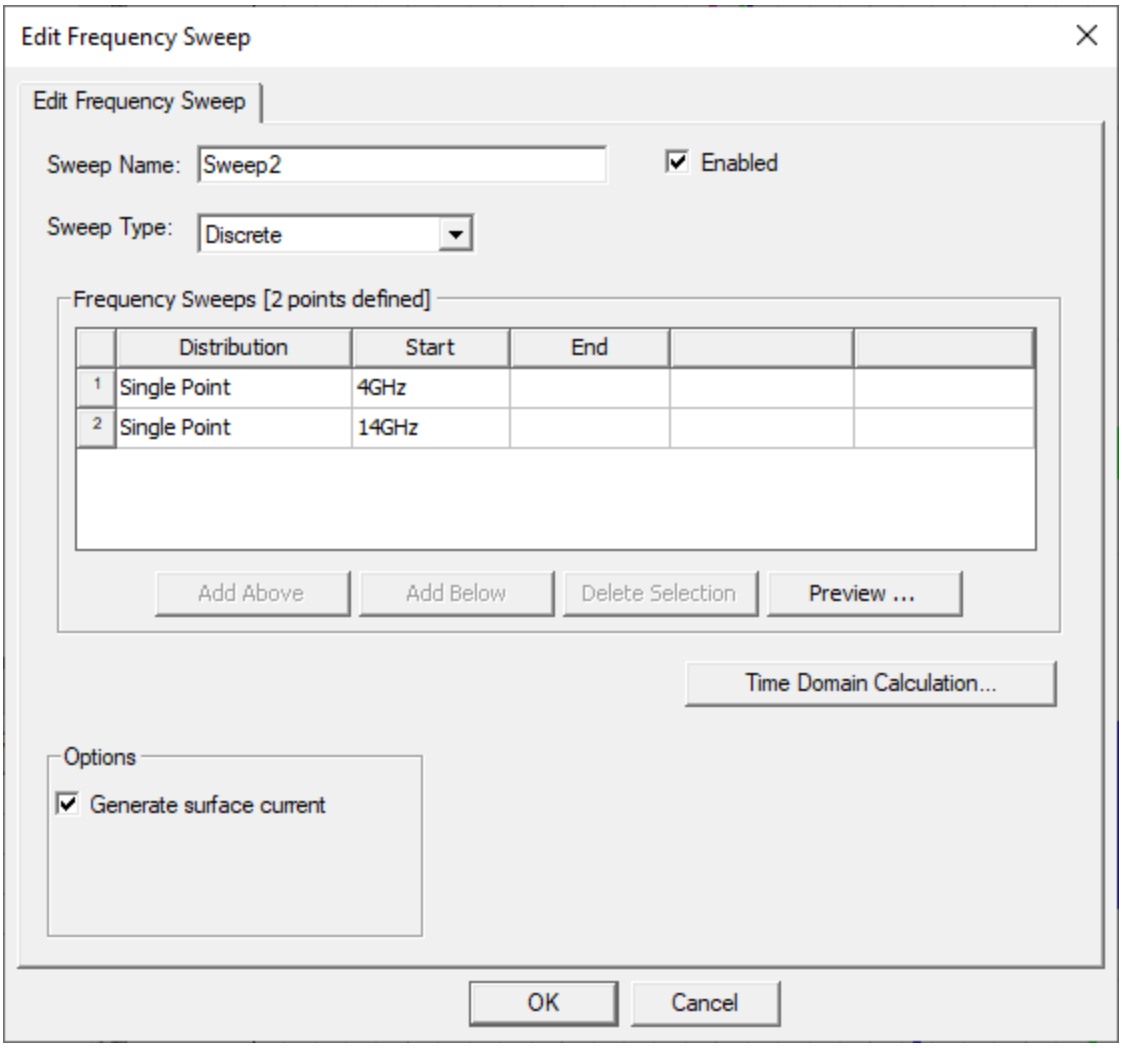
In HFSS 3D Layout designs, surface current results are only available for the last pass of an adaptive mesh or for discrete sweeps. Before viewing surface current results from the conducting layers, follow these steps to define a discrete sweep with results at two frequencies (one at a low pass-through frequency and one in the middle of the bandstop range) under the Planar EM analysis setup.

1. From the **Project Manager** window, right-click the Planar EM analysis setup (i.e., **PlanarEMSetup1**) and select **Add Frequency Sweep** to open the **Add Frequency Sweep** window.



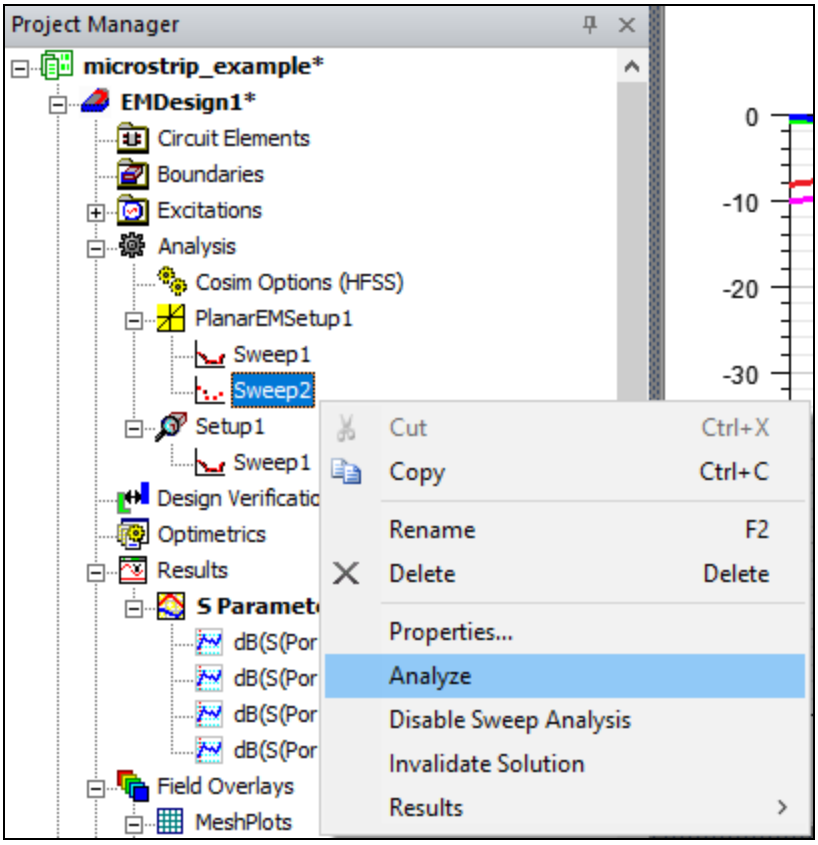
2. From the **Edit Frequency Sweep** window, do the following:
   - Select **Discrete** from the **Sweep Type** drop-down menu.
   - From the **Frequency Sweeps** table, select **Single Point** from the **Distribution** drop-down menu.
   - Enter **4** (GHz) in the **Start** field.
   - Click **Add Below**.

- In the new row, enter **14** (GHz) in the **Start** column.
- From the **Options** area, check the **Generate surface current** box.



3. Click **OK** to add the discrete sweep and close the **Edit Frequency Sweep** window.

4. From the **Project Manager** window, expand **PlanarEMSetup1** and right-click the chosen sweep (e.g., **Sweep2**). Then select **Analyze**.

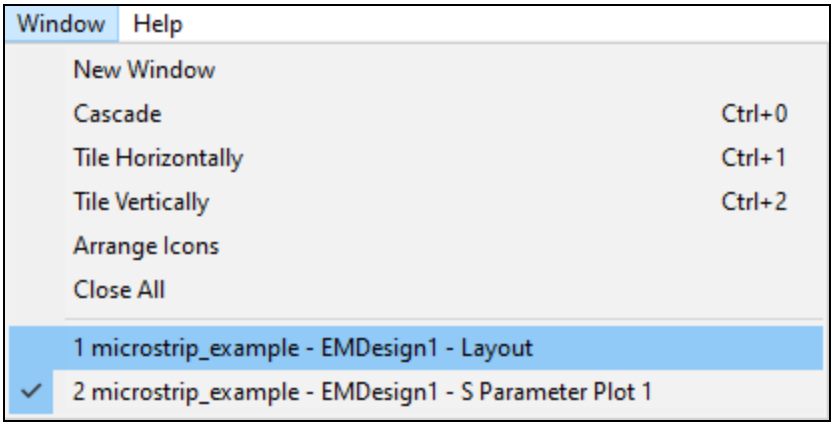


Continue to [Creating and Animating the Current Overlay].

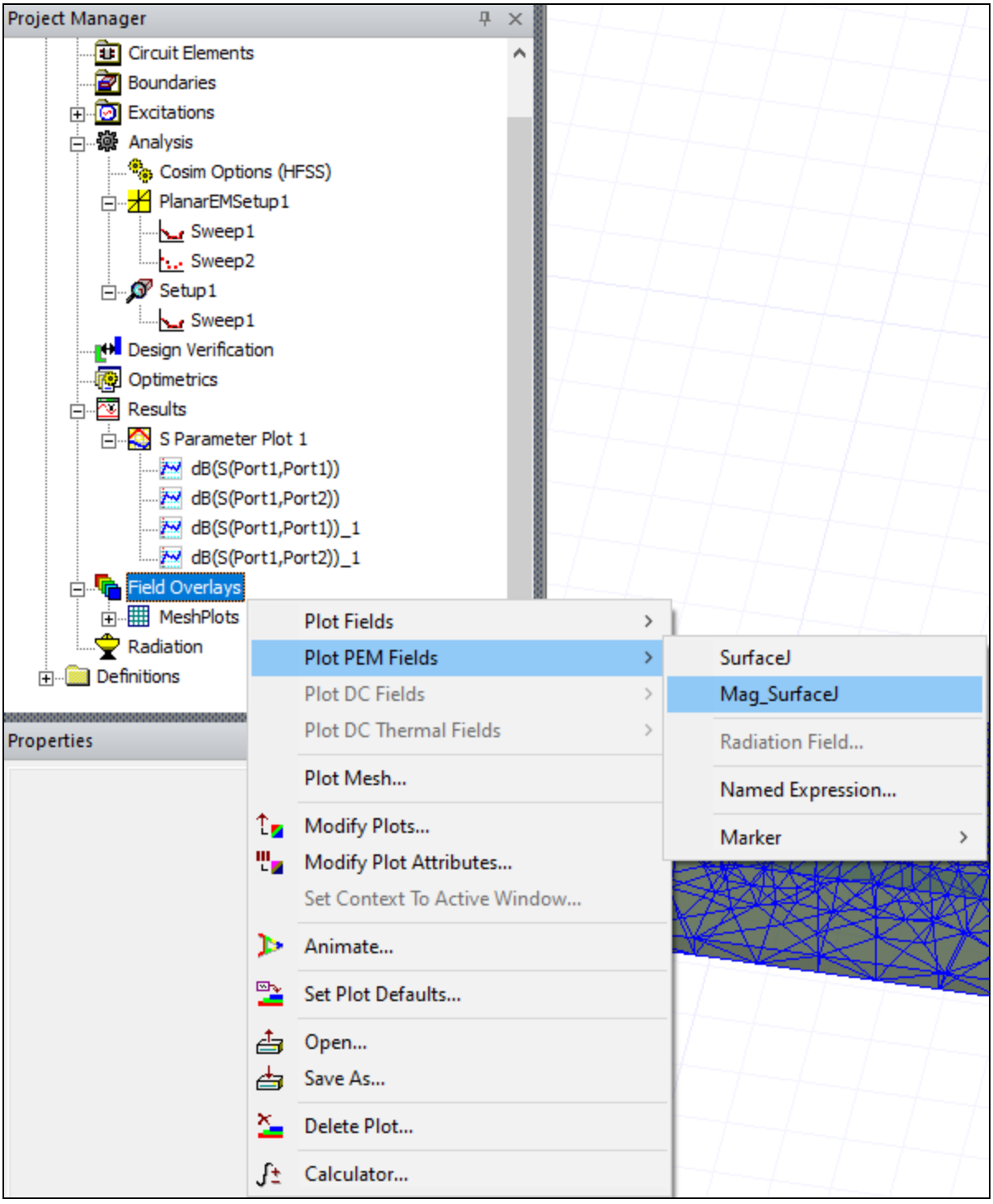
Creating and Animating the Current Overlay

Complete these steps to create and animate the current overlay.

1. From the **Window** menu, select **Layout** (i.e., **1 microstrip_example - EMDesign1 - Layout**) to return to the **Layout Editor**.



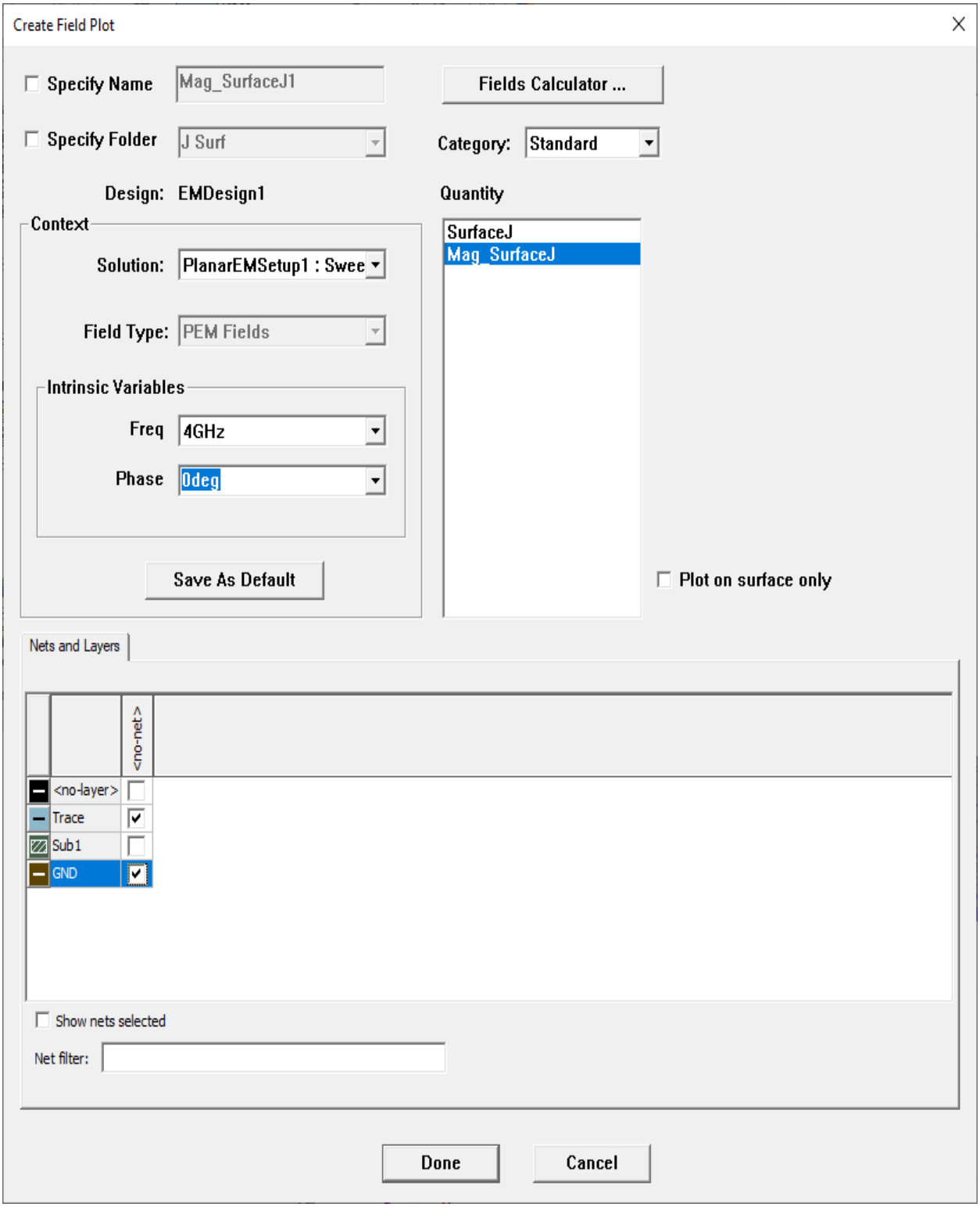
2. From the **Project Manager** window, right-click **Field Overlays** and select **Plot PEM Fields** > **Mag_SurfaceJ** to open the **Create Field Plot** window.



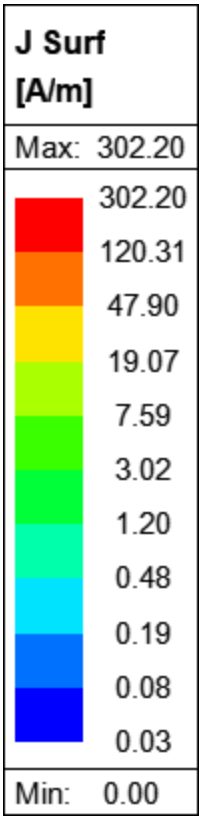
3. From the **Create Field Plot** window, ensure the following settings were selected by default:

- **PlanarEMSetup1 : Sweep2** is selected from the **Solution** drop-down menu.
- **4GHz** is selected from the **Freq** drop-down menu in the **Intrinsic Variables** area.
- **Mag_SurfaceJ** is selected from the **Quantity** list.

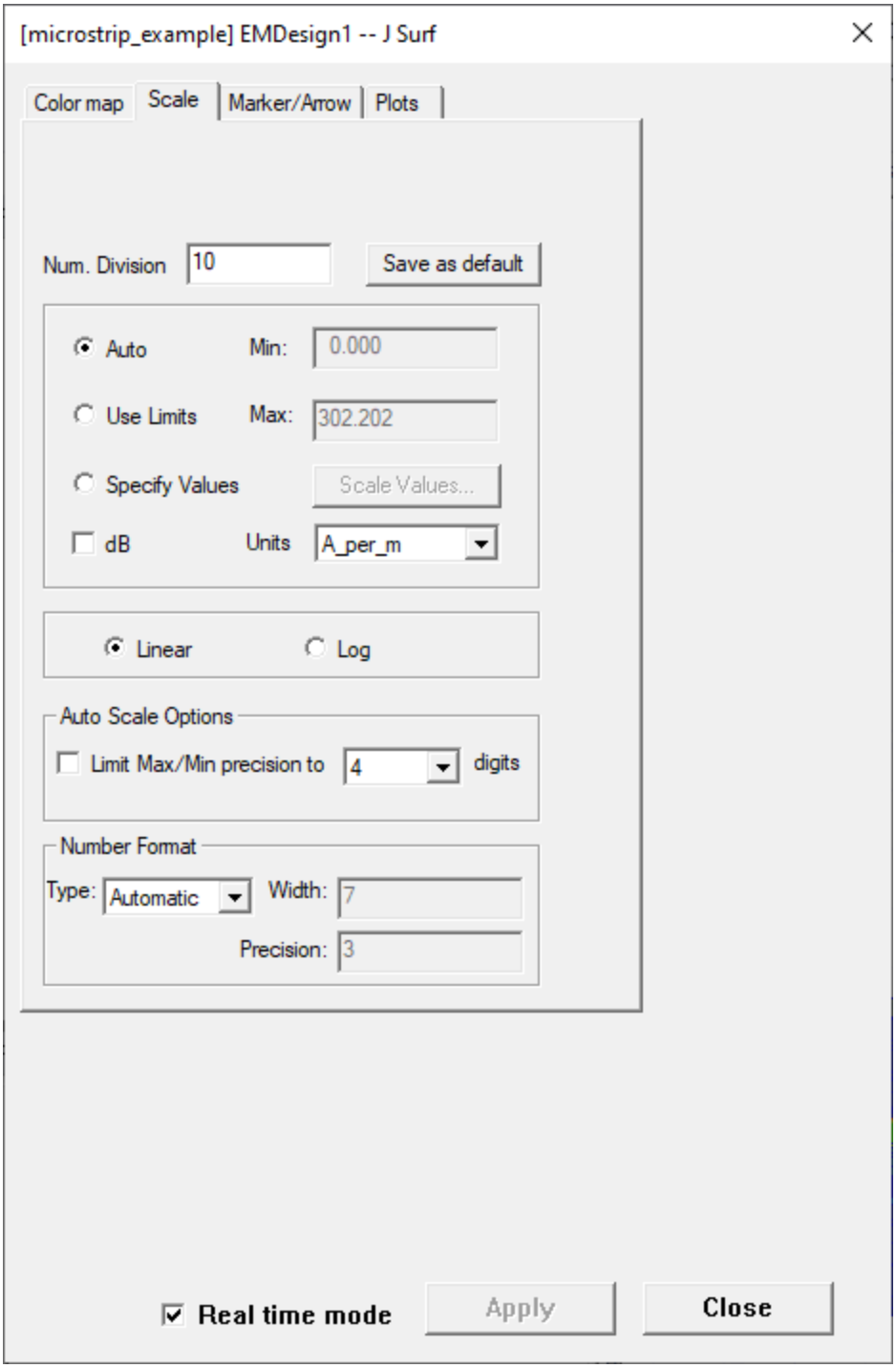
4. Under the **Nets and Layers** tab, check the **Trace** and **GND** boxes.



5. Click **Done** to close the **Create Field Plot** window and view the **J Surf** plot from the **Layout Editor**.
6. Double-click the **J Surf [A/m]** plot legend to access the plot settings.

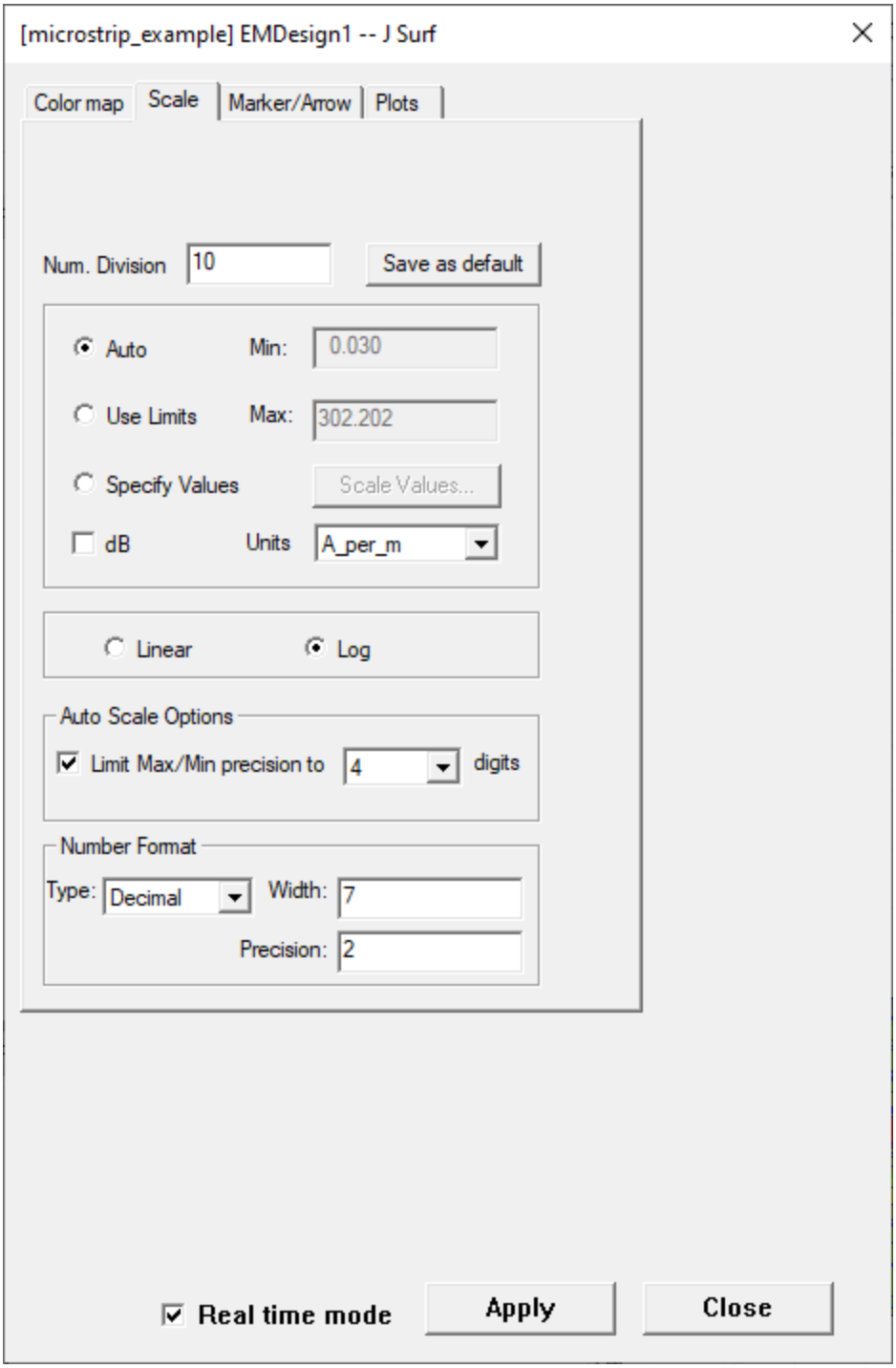


7. Navigate to the **Scale** tab.

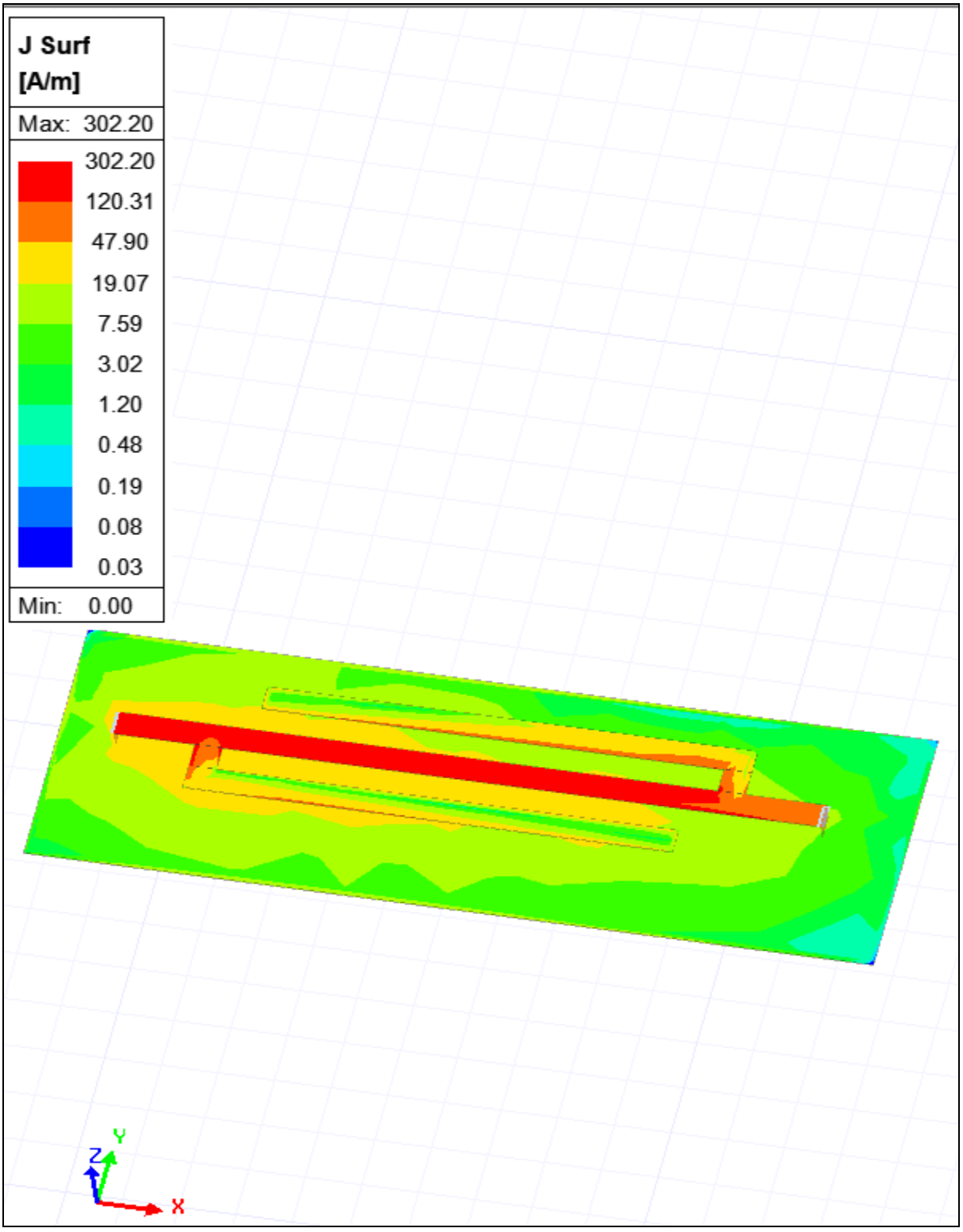


8. Ensure **10** is entered in the **Num. Division** field.

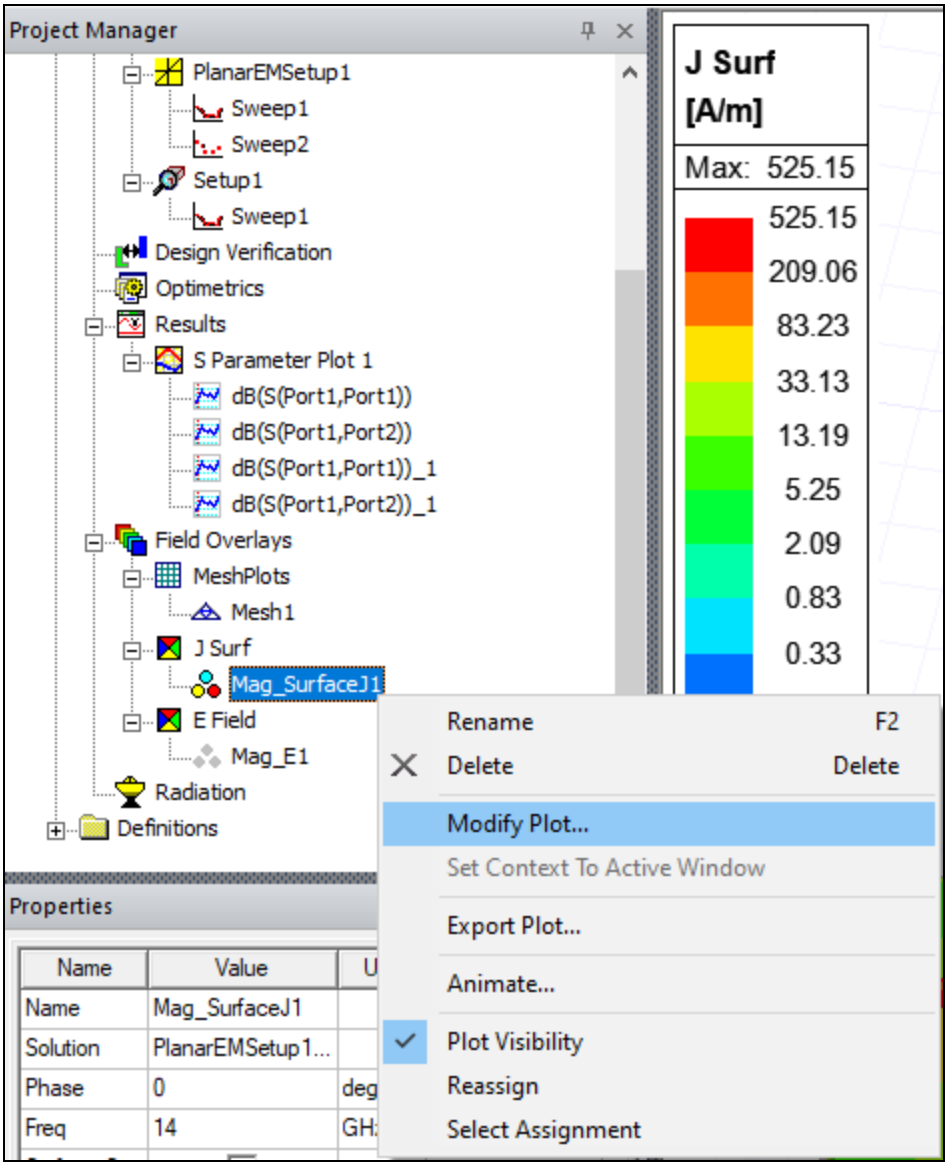
9. Make the following changes:
    - Select **Log** for a logarithmic scale.
    - From the **Number Format** area, select **Decimal** from the **Type** drop-down menu.
    - Enter **2** in the **Precision** field.



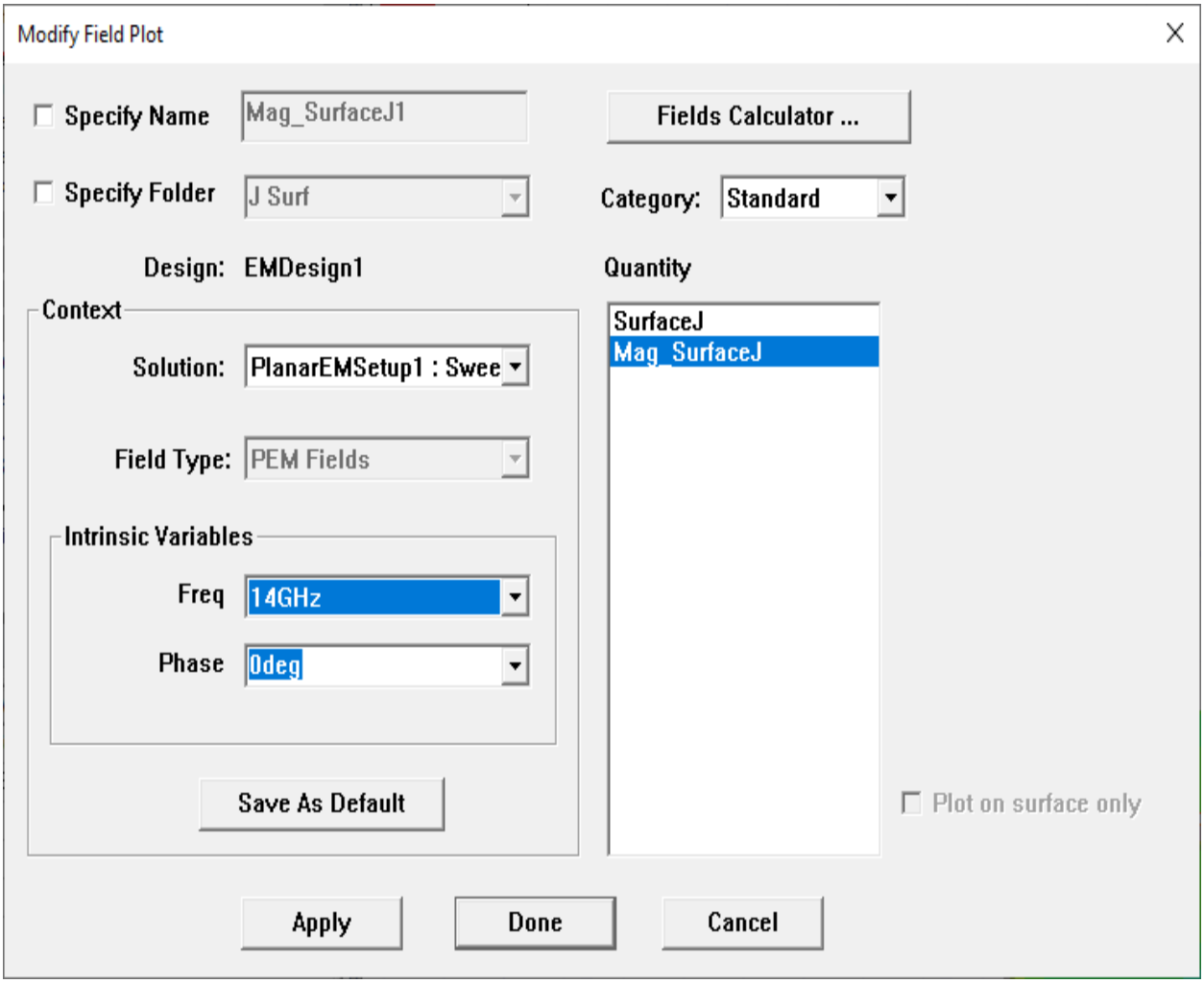
10. Click **Apply** to save changes. Then click **Close** to return to the **J Surf** plot at the **Layout Editor**.
11. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.



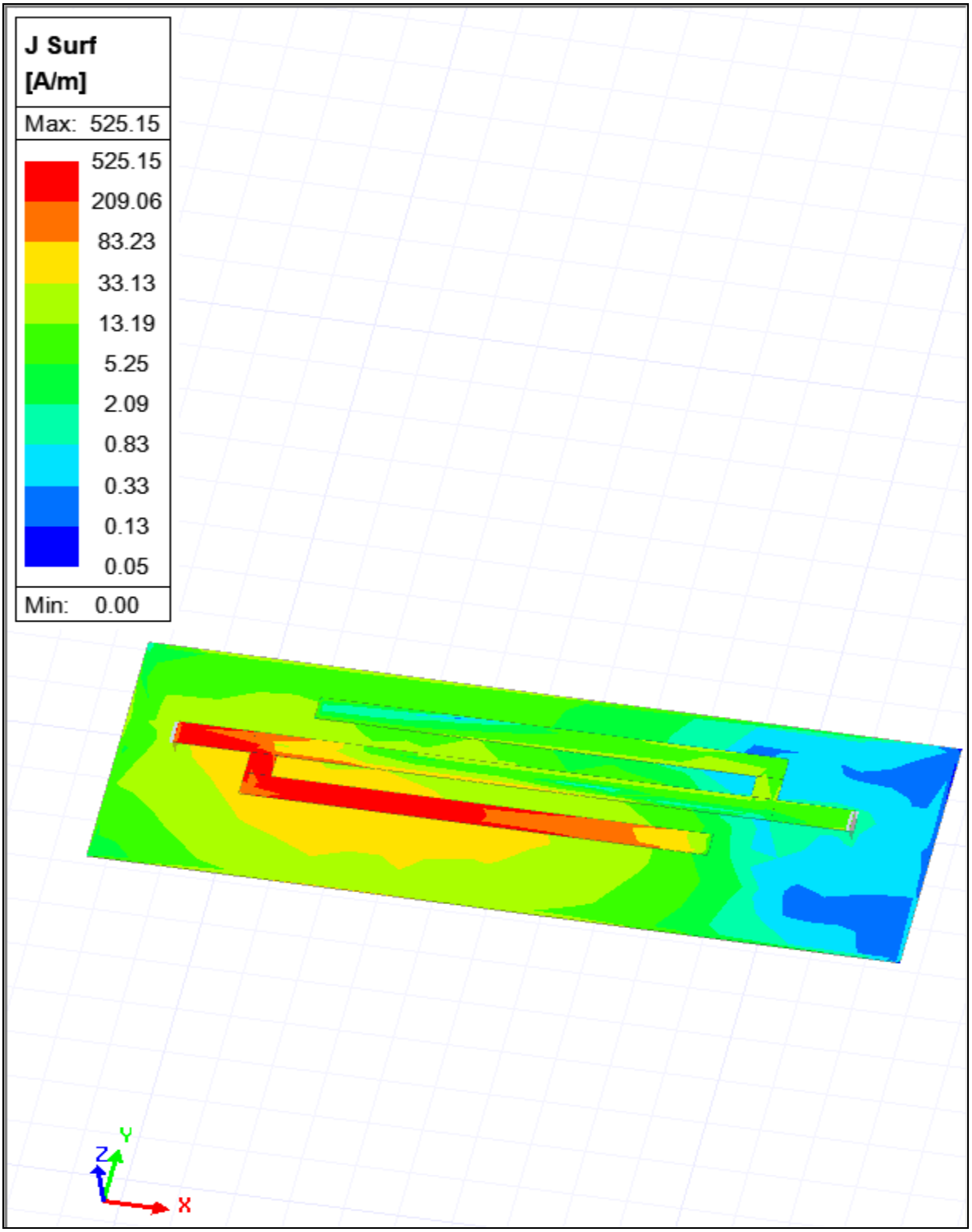
12. From the **Project Manager** window, expand **J Surf**. Then right-click **Mag_SurfaceJ1** and select **Modify Plot** to open the **Modify Field Plot** window.



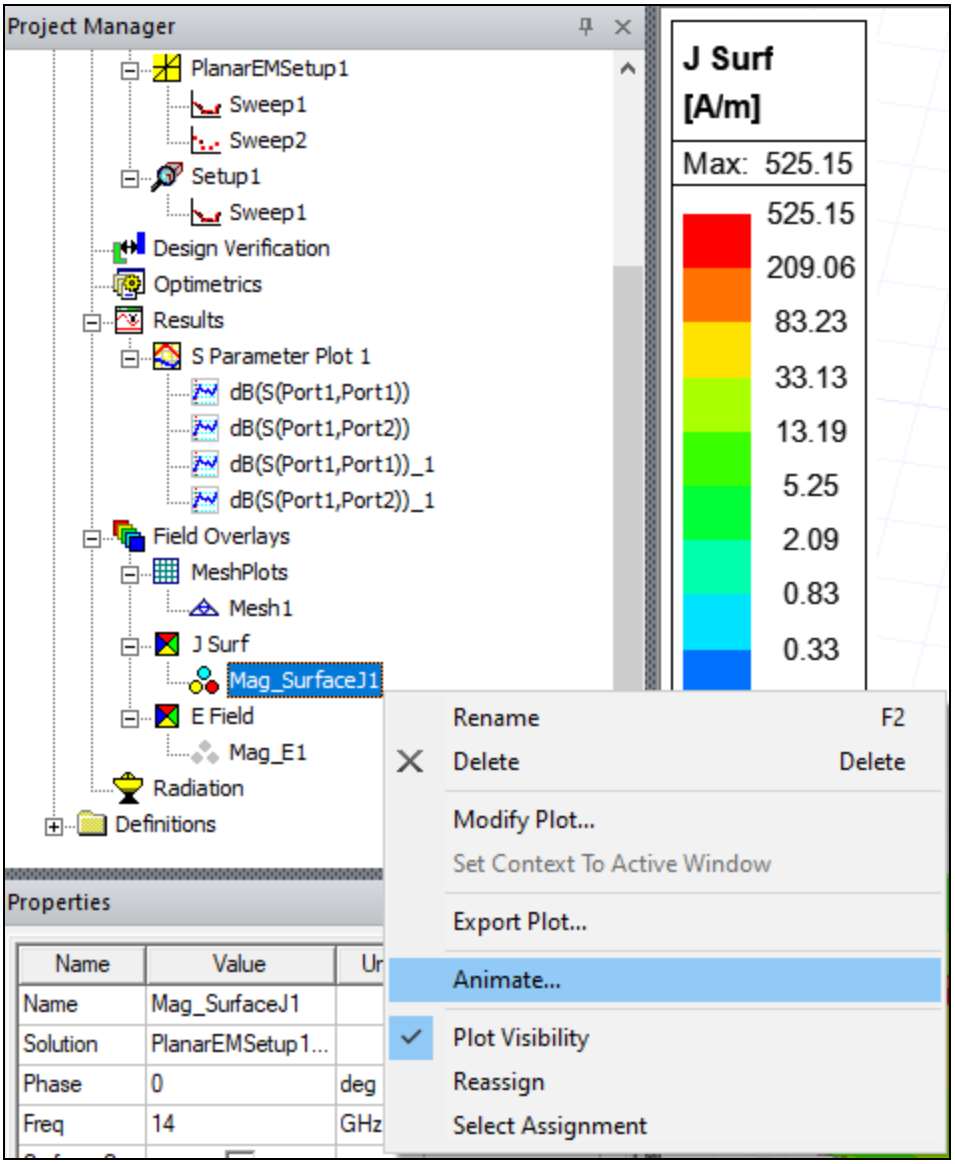
13. Select **14GHz** from the **Freq** drop-down menu in the **Intrinsic Variables** area.



14. Click **Apply**. Then click **Done** to close the window and return to the **Layout Editor**.

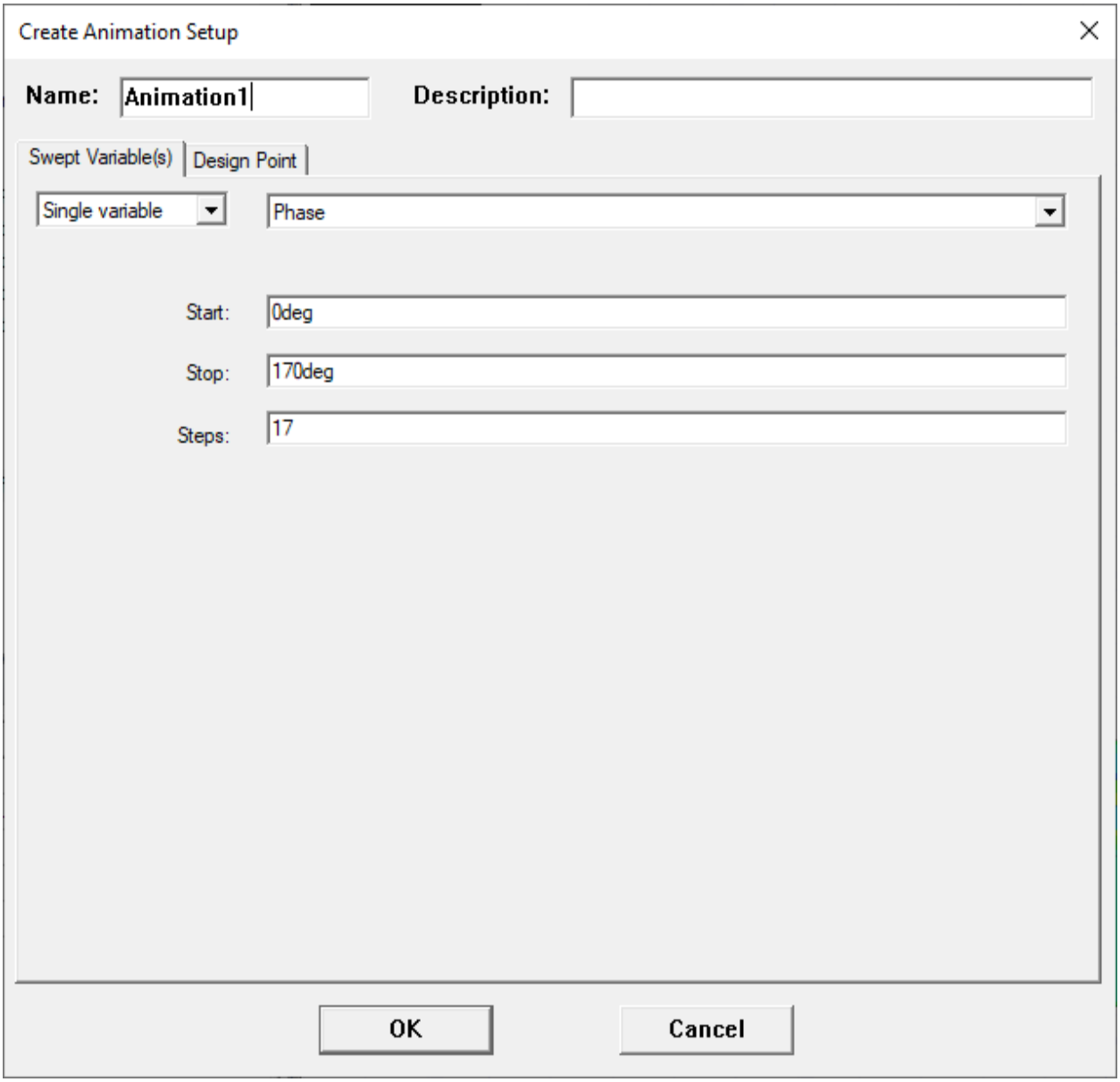


15. From the **Project Manager** window, right-click **Mag_SurfaceJ1** and select **Animate** to open the **Create Animation Setup** window.

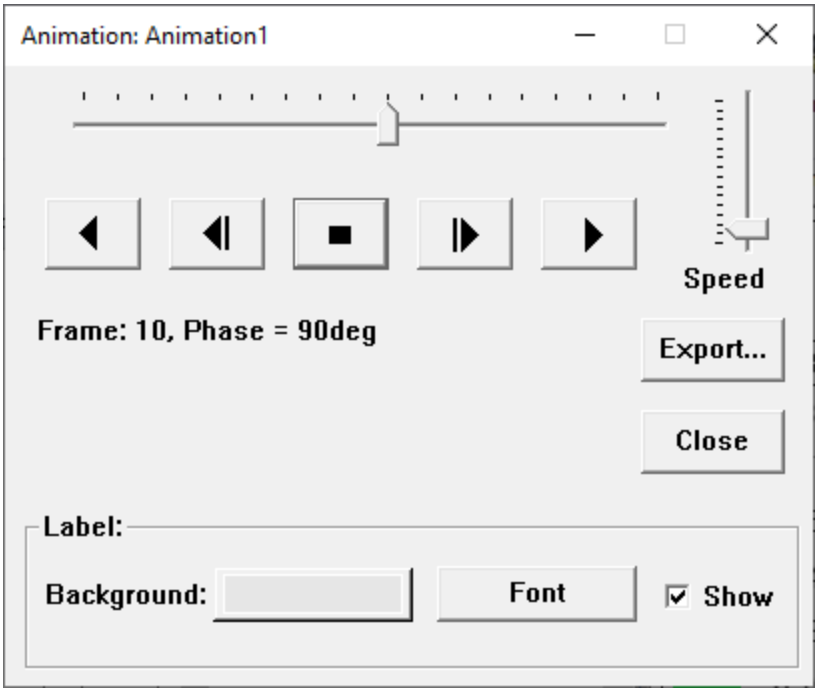


16. From the **Create Animation Setup** window, ensure the following settings were selected by default:

    - **Single variable** and **Phase** are selected from the drop-down menus.
    - **0deg** is entered in the **Start** field.
    - **170deg** is entered in the **Stop** field.
    - **17** is entered in the **Steps** field.

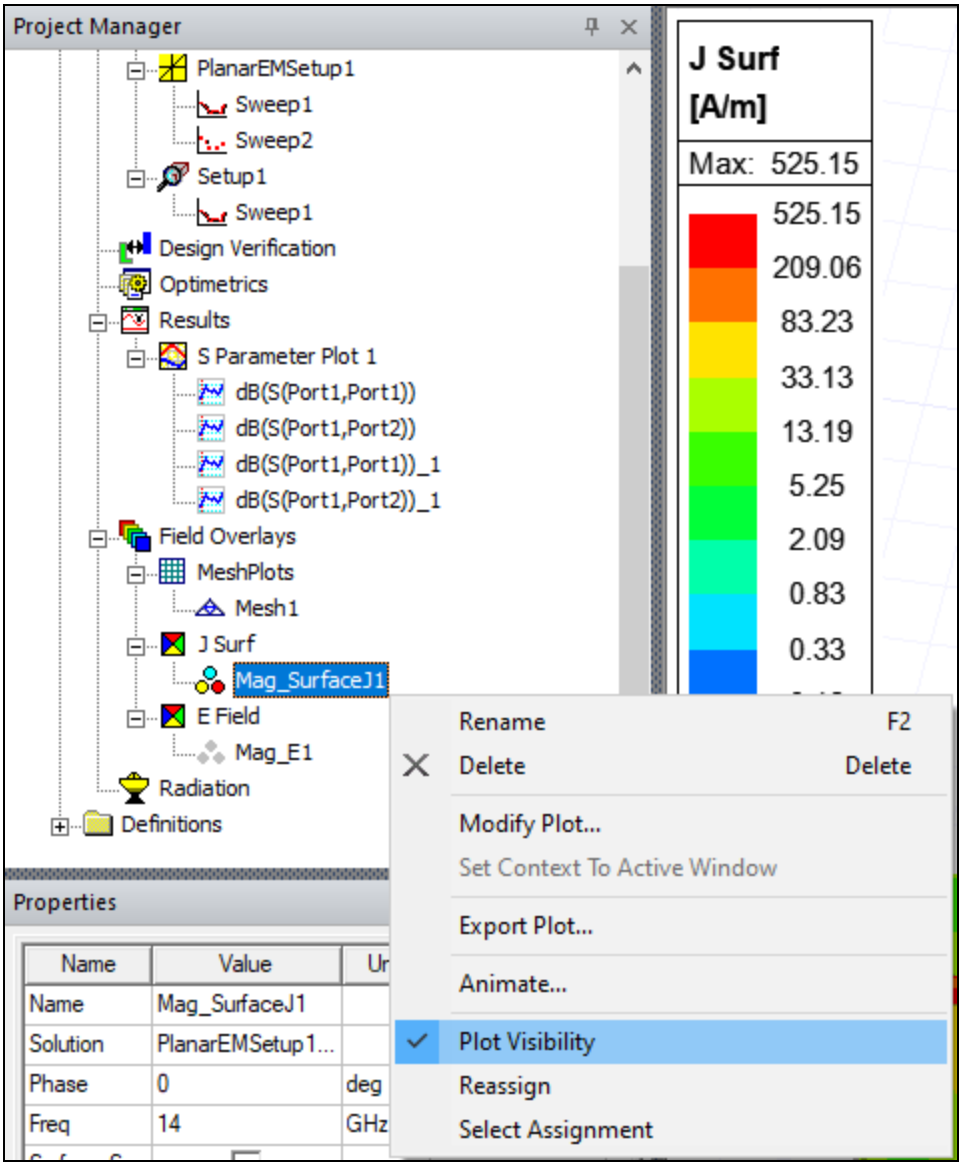


17. Click **OK** to close the **Create Animation Setup** window, open an animation control panel, and start the animation in the **Layout Editor**.



18. Use the animation controls to reverse, stop, and change the speed of the animation, among other settings.
19. From the animation control panel, click **Close** to end the animation.

20. Before continuing, navigate to the **Project Manager** window. Then right-click **Mag_SurfaceJ1** and select **Plot Visibility** to remove the adjacent check mark and hide the overlay.

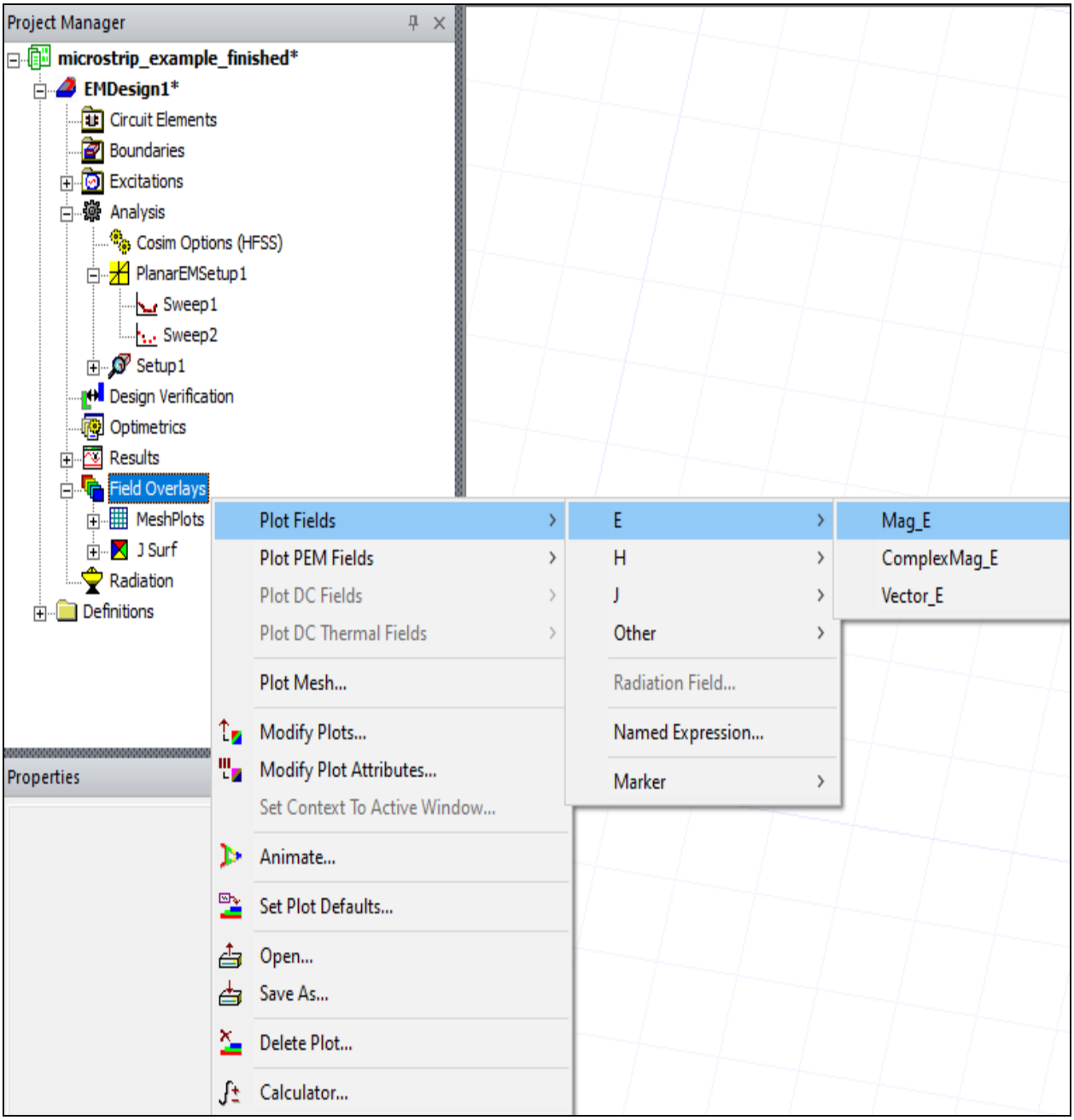


Continue to [Creating and Animating an E Field Overlay].

Creating and Animating an E Field Overlay

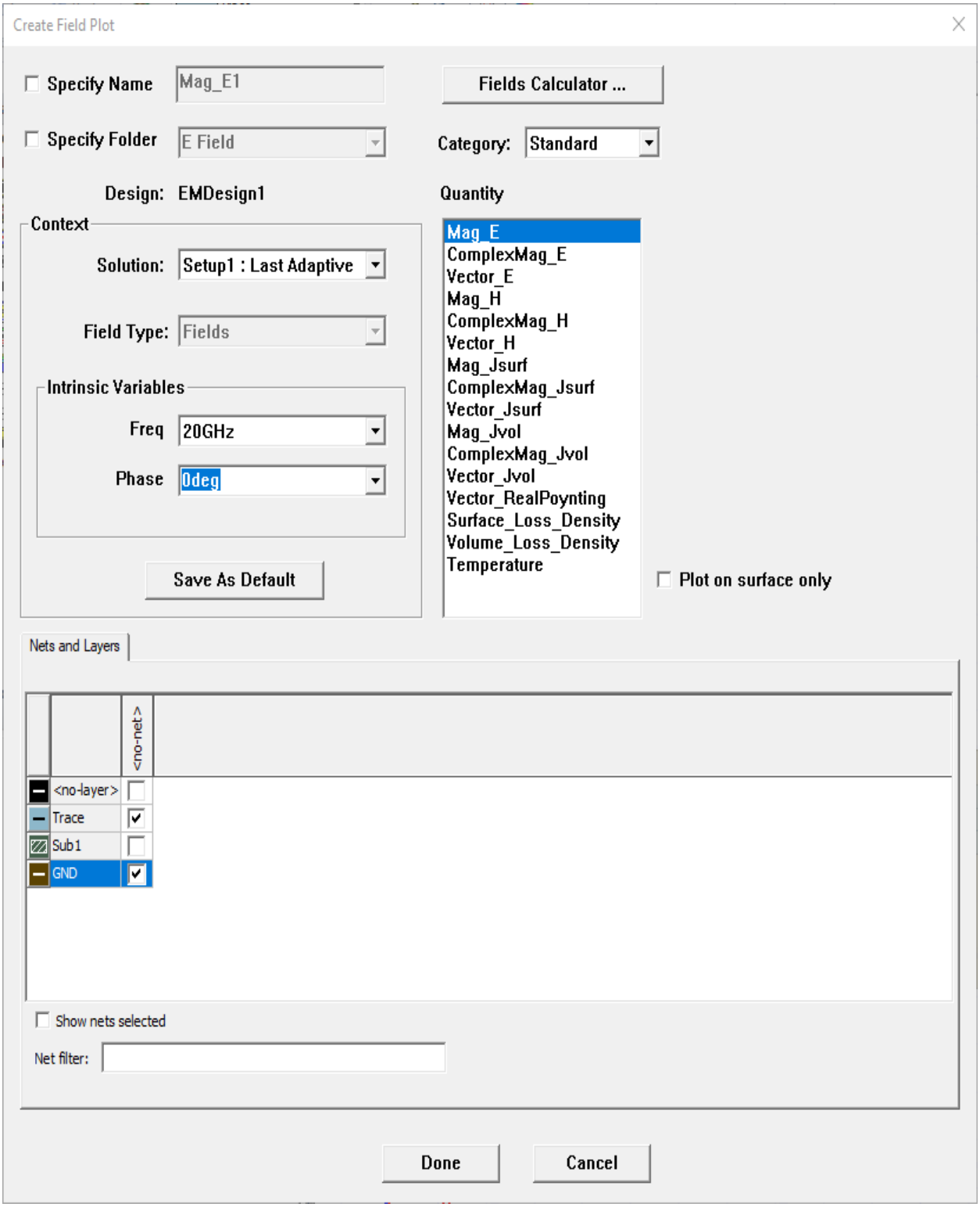
Complete these steps to create and animate the an E Field overlay.

1. From the **Project Manager** window, right-click **Field Overlays** and select **Plot Fields** > **E** > **Mag_E** to open the **Create Field Plot** window.

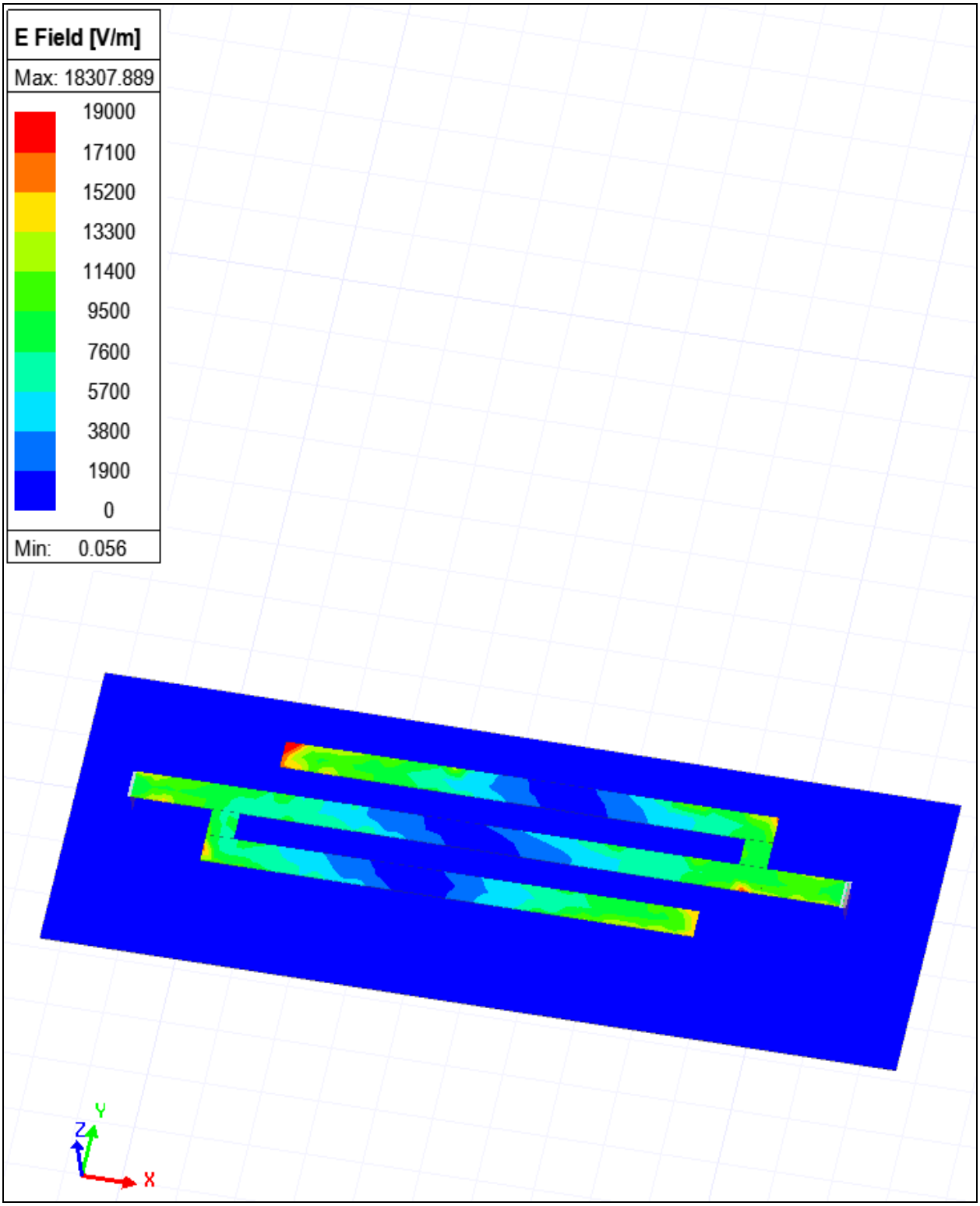


2. From the **Create Field Plot** window, ensure the following settings were selected by default:
   - **Setup1 : Last Adaptive** is selected from the **Solution** drop-down menu.
   - **0deg** is selected from the **Phase** drop-down menu in the **Intrinsic Variables** area.
   - **Mag_E** is selected from the **Quantity** list.

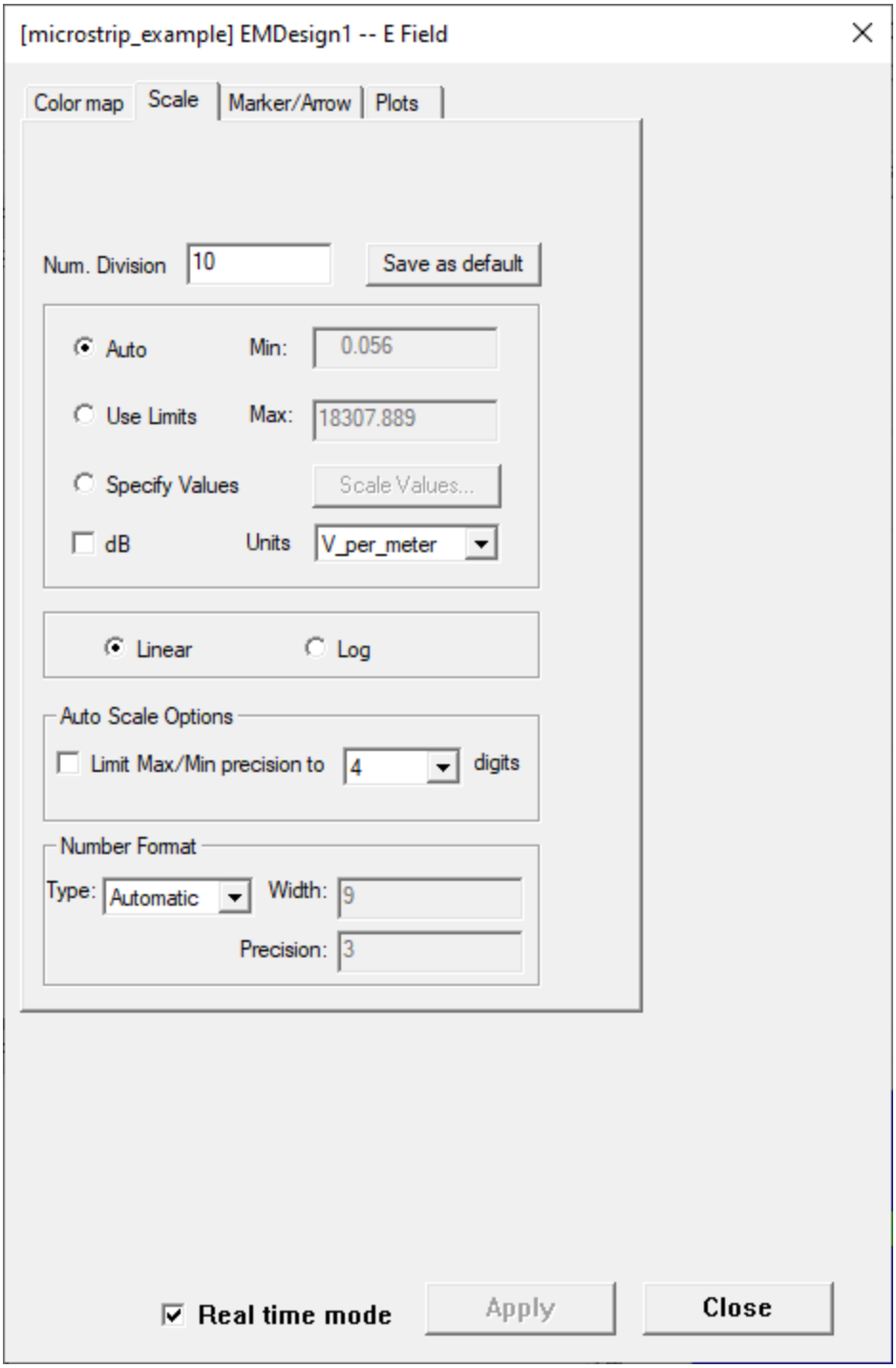
3. Under the **Nets and Layers** tab, check the **Trace** and **GND** boxes.



4. Click **Done** to close the **Create Field Plot** window and view the E Field plot from the **Layout Editor**.

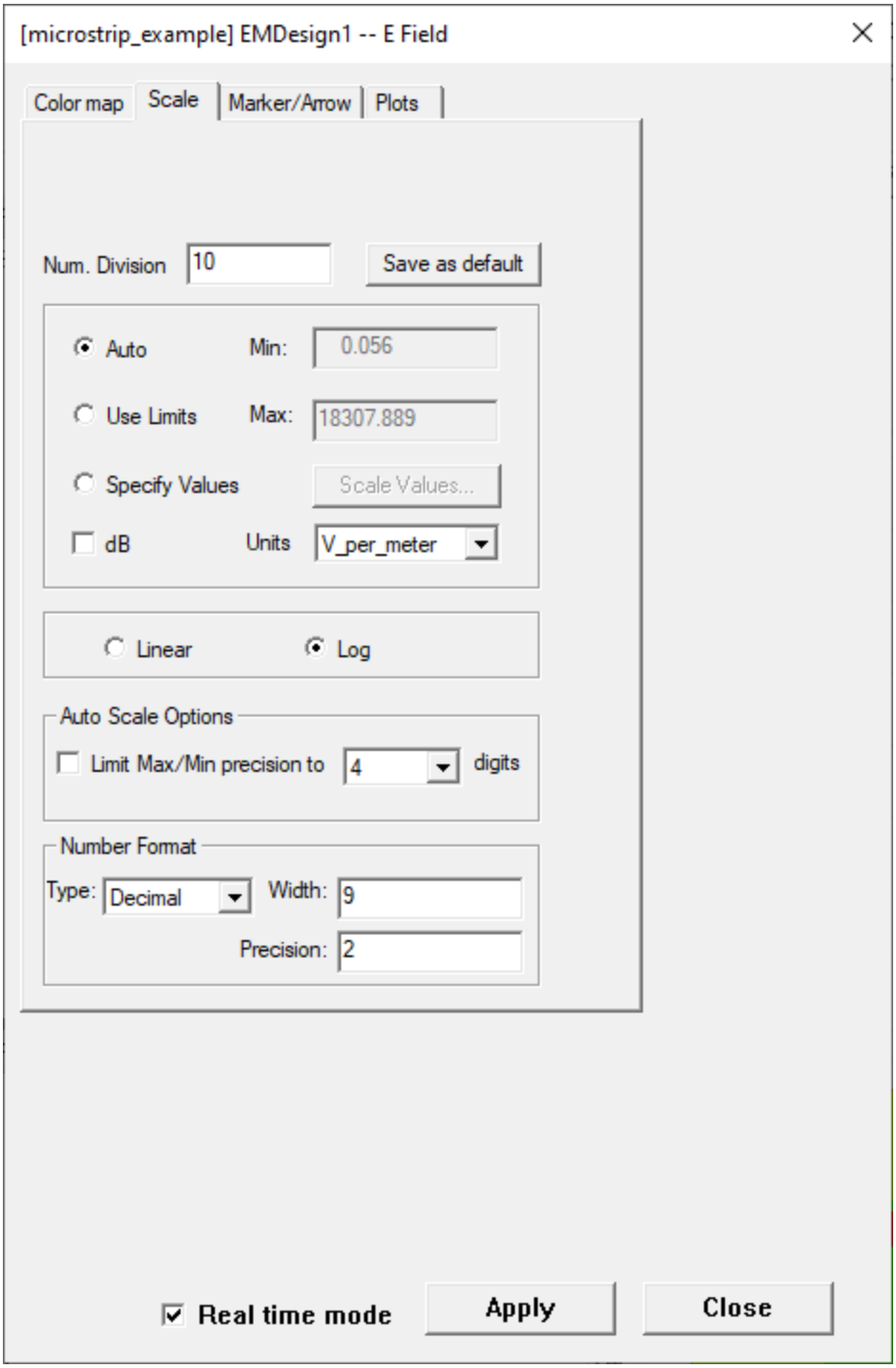


5. Double-click the **E Field [V/m]** plot legend to access the plot settings.
6. Navigate to the **Scale** tab.

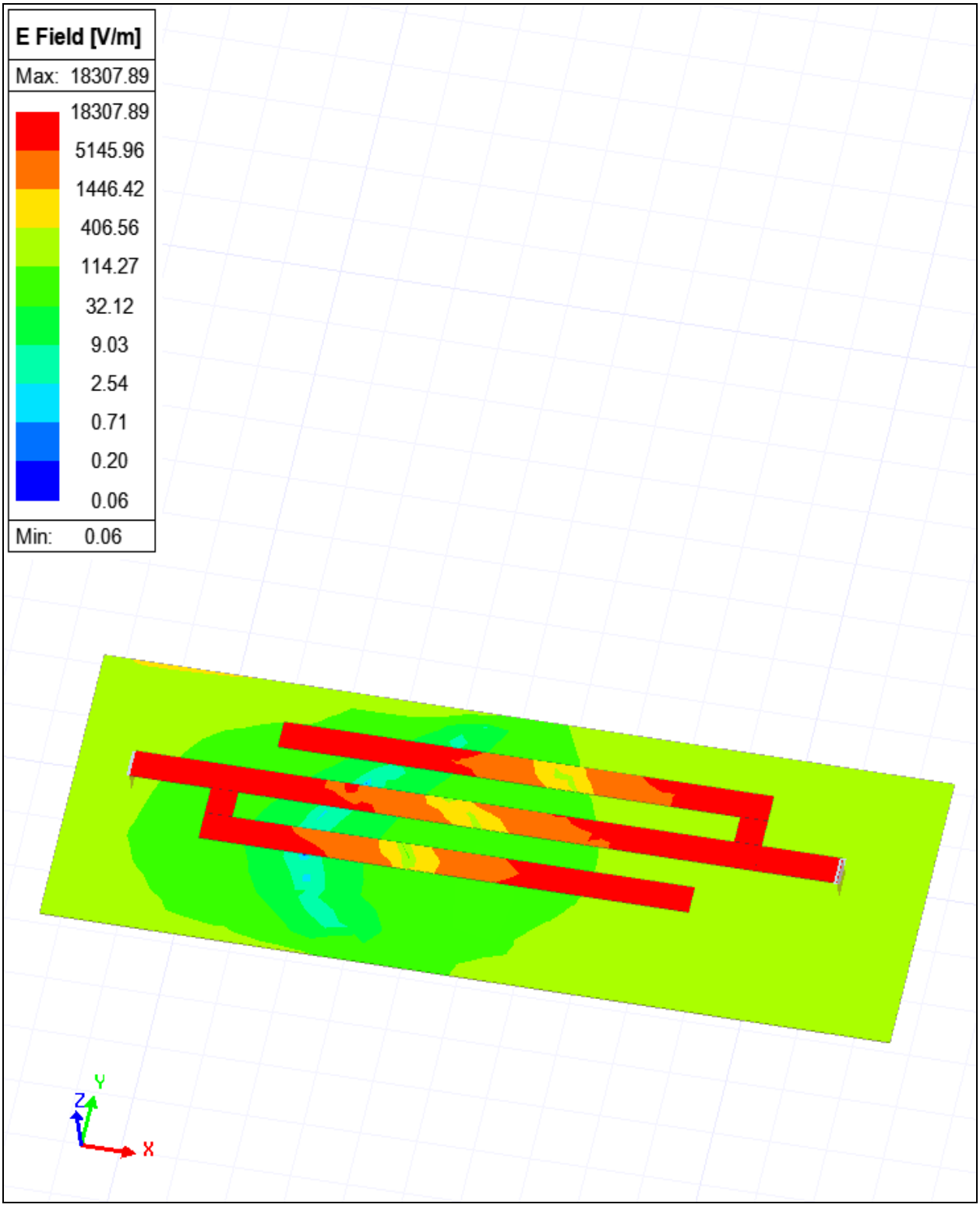


7. Ensure **10** is entered in the **Num. Division** field.

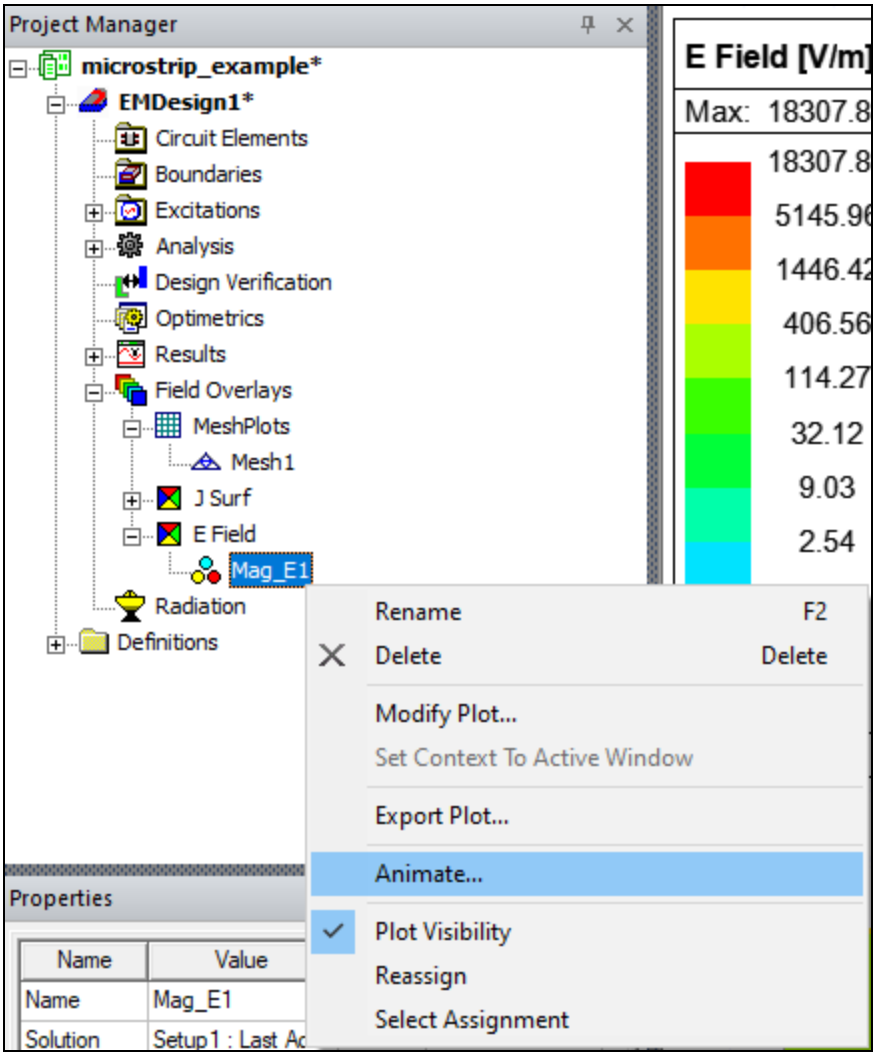
8. Make the following changes:
    - Select **Log** for a logarithmic scale.
    - From the **Number Format** area, select **Decimal** from the **Type** drop-down menu.
    - Enter **2** in the **Precision** field.



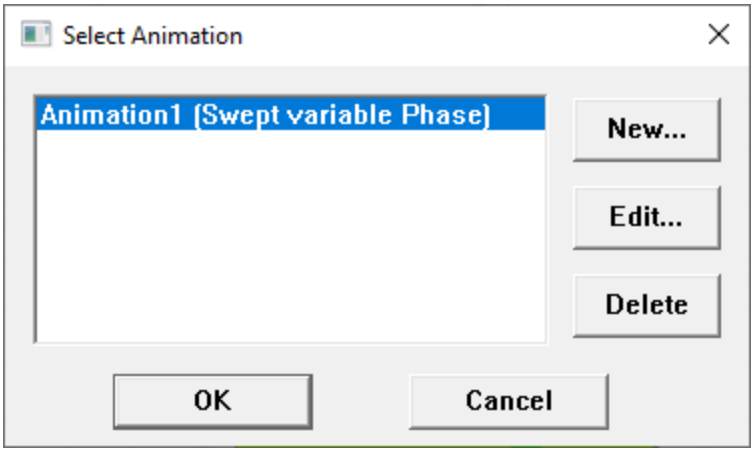
9. Click **Apply**.
10. Click **Close** to return to the E Field plot at the **Layout Editor**.
11. From the **Layout Editor**, **Zoom**, **Rotate**, or **Pan** using the standard **Layout Editor** controls.

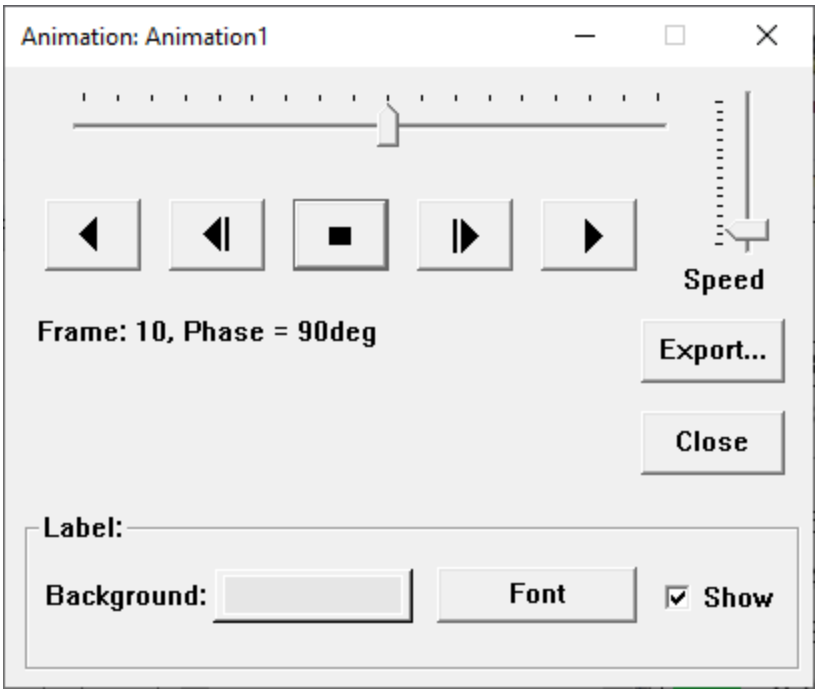


12. From the **Project Manager** window, right-click **Mag_E1** and select **Animate** to open the **Select Animation** window.



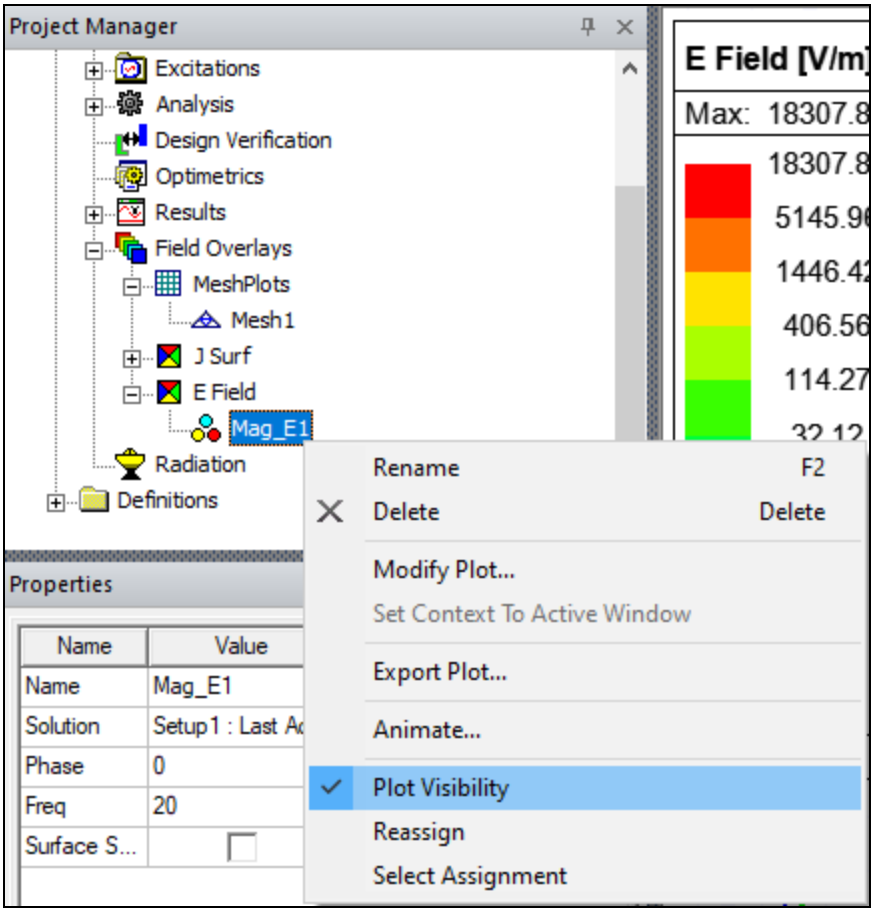
13. Click **OK** to select the only available option (i.e., **Animation1 (Swept variable Phase)**), which was defined in the previous step. The **Select Animation** window closes, an animation control panel opens, and the animation begins in the **Layout Editor**.





14. Use the animation controls to reverse, stop, and change the speed of the animation, among other settings.
15. From the animation control panel, click **Close** to end the animation.

16. From the **Project Manager** window, right-click **Mag_E1** and select **Plot Visibility** to remove the check mark and hide the mesh.

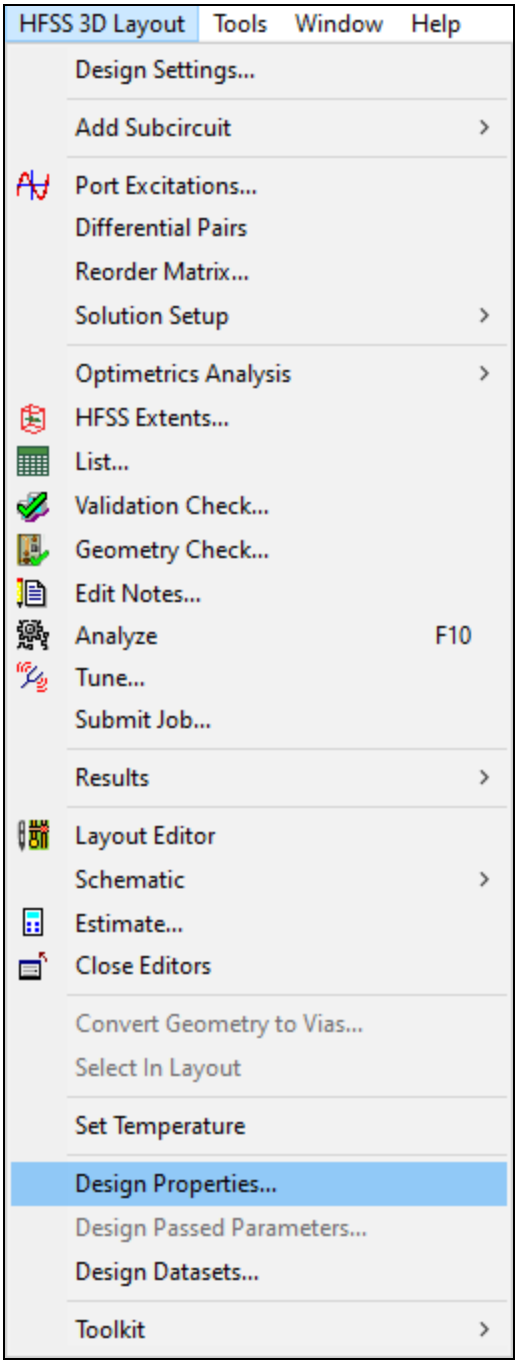


Continue to [Optional Challenge Exercise](#).

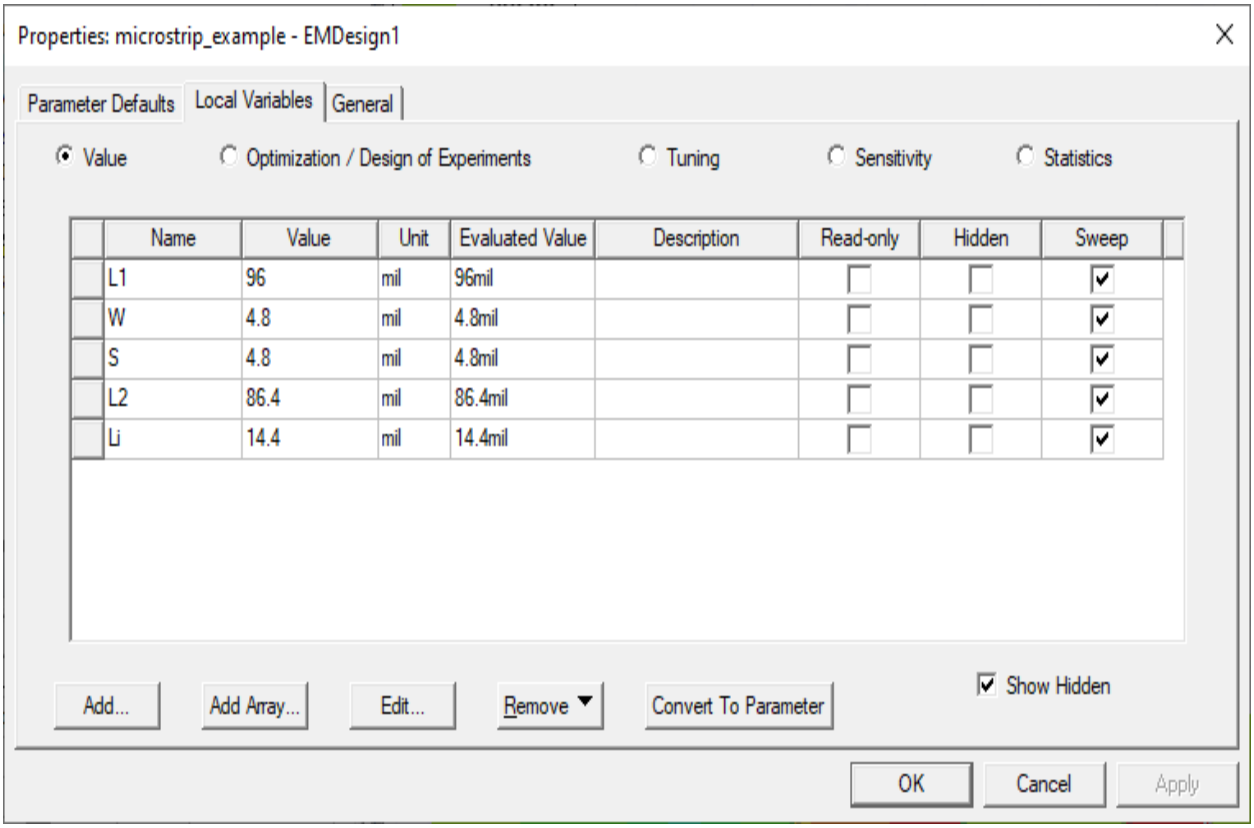
Optional Challenge Exercise: Experimenting With Parameterization

Since the geometry of this model is parameterized, experiment with how easy it is to alter the geometry and reanalyze the results. Parameterization is a convenient way to perform analyses to see the effects of different design variations. Complete these steps to locate and edit the design variables, as chosen.

1. From **HFSS 3D Layout**, select **Design Properties** to open the **Properties** window.



2. Navigate to the **Local Variables** tab.

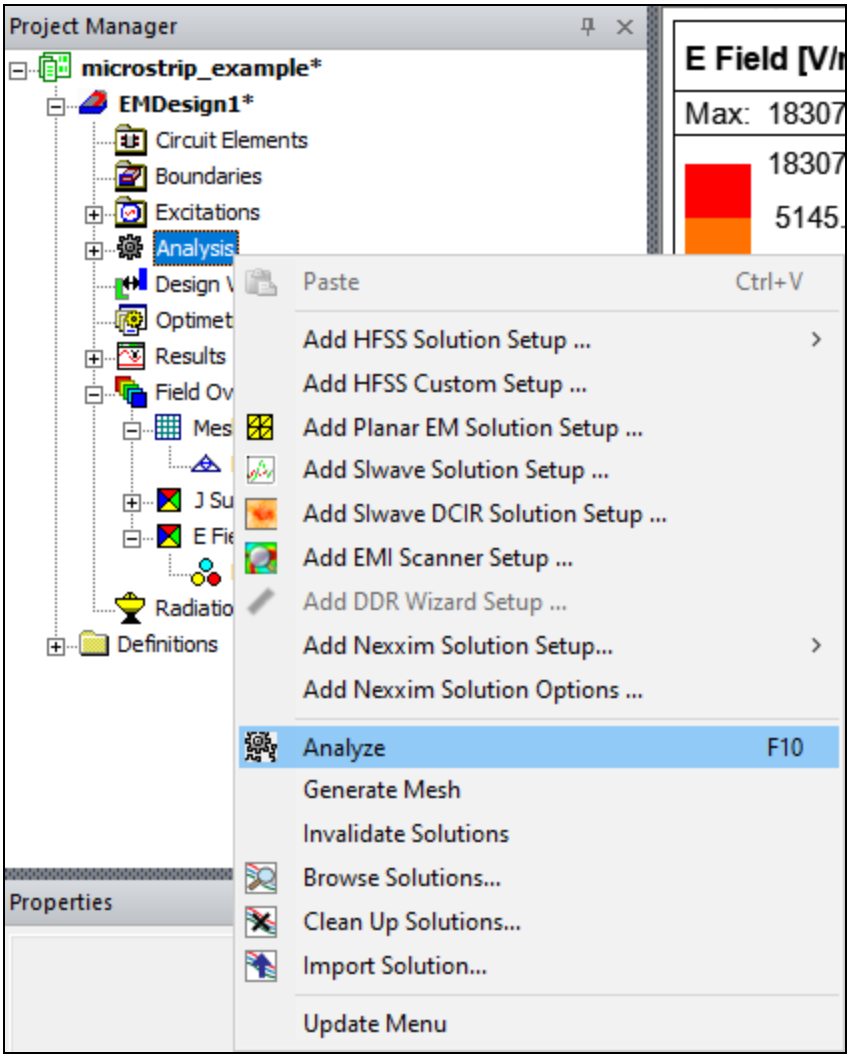


3. The **Local Variables** tab contains the variables that can be defined by the user, listed in the order in which they are defined. Enter a different number in the **Value** fields to alter the dimensions of the model, as chosen.

> **Warning:**
>
> Do not to make excessive adjustments to the variables. The ground plane is drawn at a fixed size and the trace objects must remain in the perimeter of the ground plane.

4. Click **Apply** to update the model in the **Layout Editor**.
5. Click **OK** to close the **Properties** window.

6. From the **Project Manager** window, right-click **Analysis** and select **Analyze** to rerun all analysis setups of the altered model.



7. Observe any changes in the S-parameter plot once the solution is finished.

> **Note:**
>
> The user can also automate the selection of design property values to achieve targeted results. For more information, search for "*optimization*" or "*design of experiments*" in the Help.

Congratulations, the HFSS 3D Layout microstrip filter getting started guide is complete.